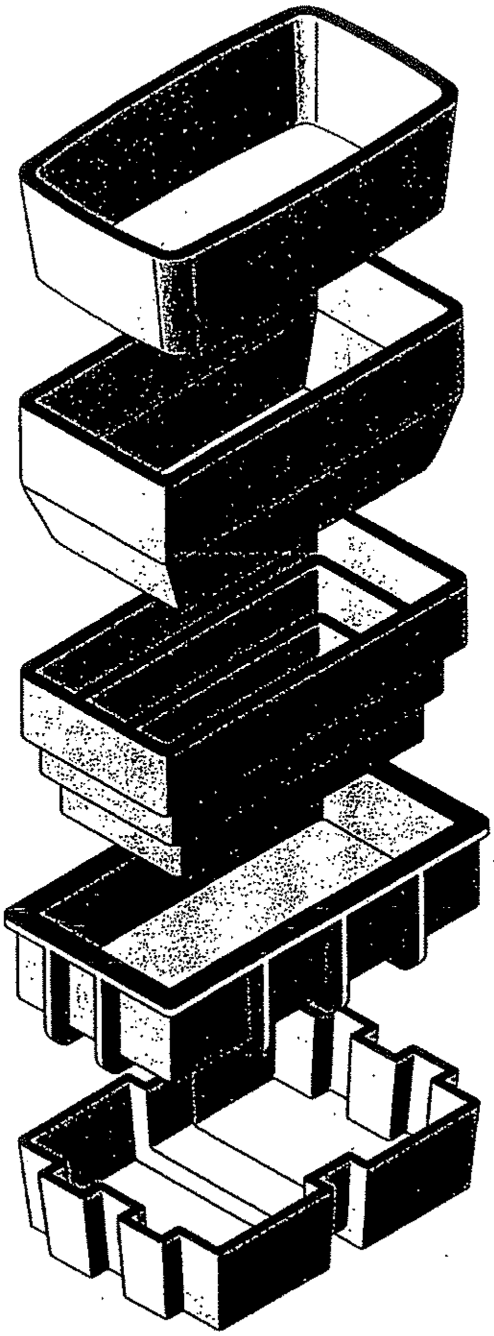
Figure 2-2

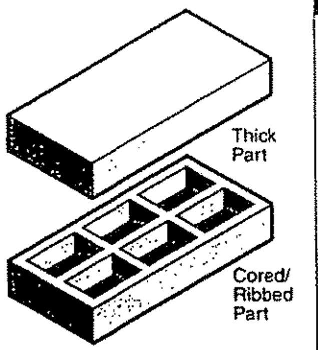


Part-stiffening techniques.

- If you double wall thickness in a flat part, the part's stiffness will increase by a factor of eight.
- Wall thickness for solid materials is typically 1/8 inch, although parts with walls as thick as 1/2 inch have been molded successfully.
- Wall thickness for Baydur STR/C or STR/F composite materials typically ranges from 1/16 to 1/4 inch, although parts with thicker walls have been molded.
- Another method to stiffen a side wall in the direction of draw is to curve it at the base, or redesign the flat section so that it has steps, angles, or corrugations (see figure 2-2).
- Wall thickness for parts made of Baydur structural foam can range from 1/4 to 1-1/2 inches.

Because a part's thickest cross section determines molding time, excessively thick cross sections may cause uneconomical and long molding cycles. Thin-walled parts have the shorter mold-cycle times, because the heat of reaction dissipates more rapidly.

Figure 2-3



Cored part.

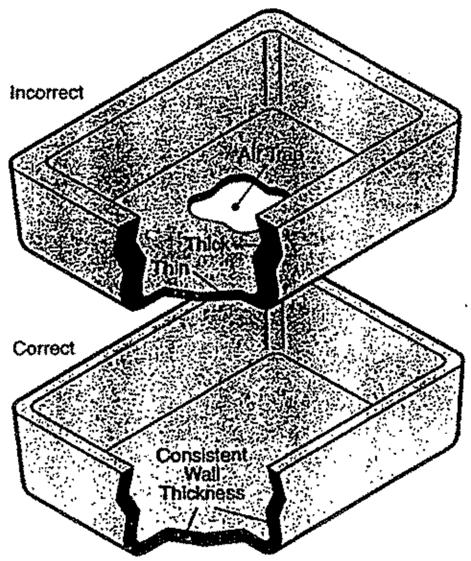
While RIM polyurethane systems can be used to make parts with varying wall thicknesses, designing parts with excessive wall-thickness variations may cause uneven filling and racetracking. Figure 2-4, showing a five-sided box, is a good example of racetracking. The liquid components fill the thicker walls, leaving air entrapments in the thinner base. To correct for this racetracking effect, design thinner side walls or a thicker base.

Unusually thick cross sections can also cause dimensional difficulties. Because the material in thick cross sections takes longer to cool, parts may shrink more and can possibly warp. In extreme cases, scorching or splitting may occur. Whenever possible, core thick sections to avoid this effect (see figure 2-3). Consider using ribs or other local reinforcements to increase part stiffness as an alternative.

RIB DESIGN AND CONFIGURATION

Taller, thinner ribs are more effective than shorter, wider ones (see figure 2-5). In this figure, both ribs have the same cross-sectional area, but the stiffening effect of rib B is far greater than that of rib A. Ribs should run continuously from side-to-side or corner-to-corner whenever possible. The lowest rib height determines the effective stiffness of notched ribs (see figure 2-6).

Racetracking Figure 2-4



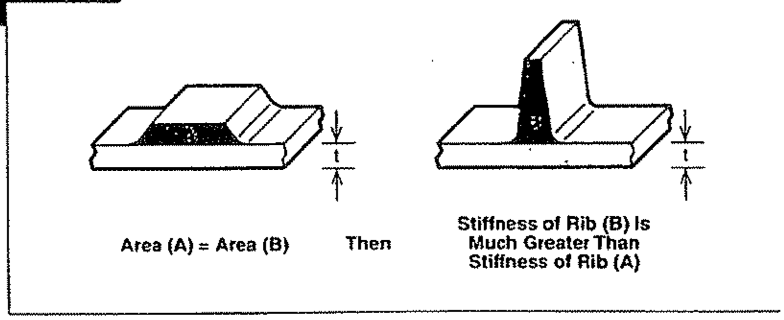
CW 0308696

Taller ribs with draft may lead to a wide base, resulting in problems in processing, with cycle time, and with product appearance.

Ribs and other protrusions that are thicker than the nominal wall can cause "read through" — sink marks or visual blemishes on the opposite show surface (see figure 2-7). In general, sink marks are much less of a problem with RIM polyurethanes than with thermoplastics. These sinks appear where the rib and mating wall meet because the increased wall thickness leads to increased local-area shrinking as the part cools. Designing a step in the part where the rib meets the mating wall helps to avoid sink marks (see figure 2-8).

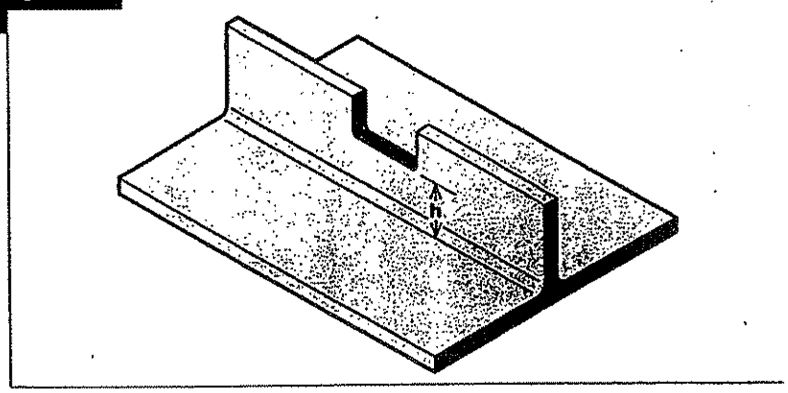
If you need the support of a thick rib, design it as a series of thinner ribs with equivalent height and cross-sectional area. The space between these thinner ribs should be no less than the nominal wall thickness (see figure 2-9).

Figure 2-5



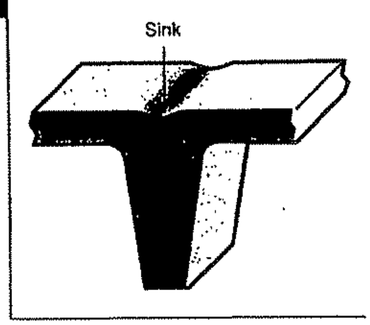
Thick versus thin ribs.

Figure 2-6



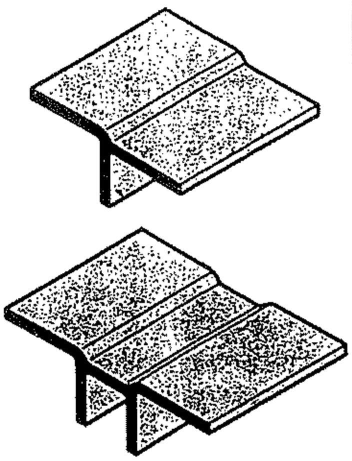
Notched rib.

Figure 2-7



Sinks caused by thick ribs.

Figure 2-8



Offset rib.

Side walls may need to be stiffened in the direction of draw and/or perpendicular to the direction of draw. For stiffening in the direction of draw, use simple ribbing. When perpendicular ribbing is necessary — such as in walls — you may have to use sliding cores, which may add significantly to mold and finishing costs. Additional design considerations for ribs include:

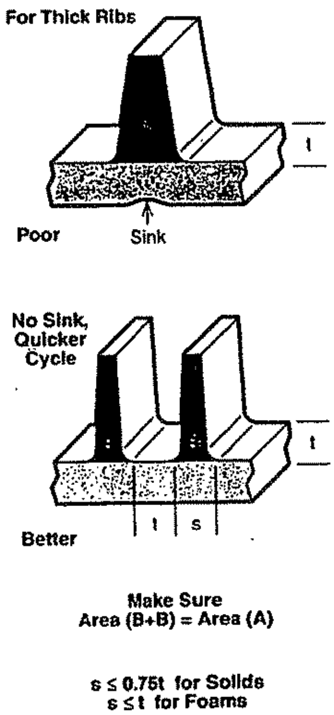
- Ribbing increases stiffness only in the ribbing direction of that rib.
- If a rib is notched, the lower section of the rib will determine strength, unless the notch is bridged with a metal stiffener (see figure 2-10).

Ribs are quite difficult to mold in parts made of composite materials. Typically these ribs may have resin-rich, potentially brittle areas at their tops, because it is difficult to get mat into this tight area.

Ribbing Direction

Figure 2-11 illustrates three common ribbing configurations. If a part needs to be reinforced in only one direction, use **parallel ribbing**. Use **bidirectional ribbing** if your part needs to be reinforced in both directions of the plane surface. Apply bidirectional ribs sparingly because excessive ribbing can make your part heavy and uneconomical to produce. Additionally, avoid placing ribs perpendicular to the anticipated flow direction, because they may trap air and cause filling difficulties (see figure 4-3).

Rib Configuration Figure 2-9



Convert thick ribs into evenly spaced, thin ribs.

Figure 2-10 Notched Rib with Bridge

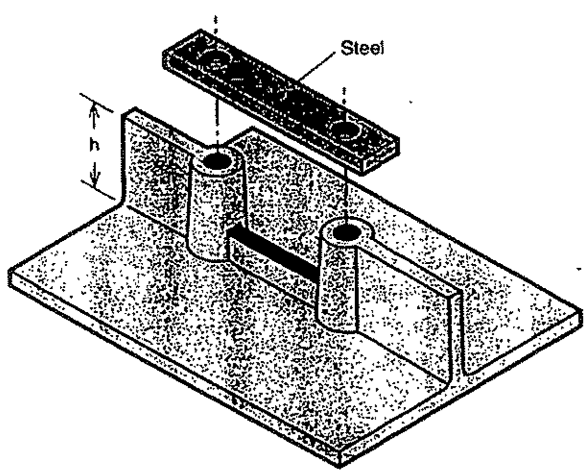
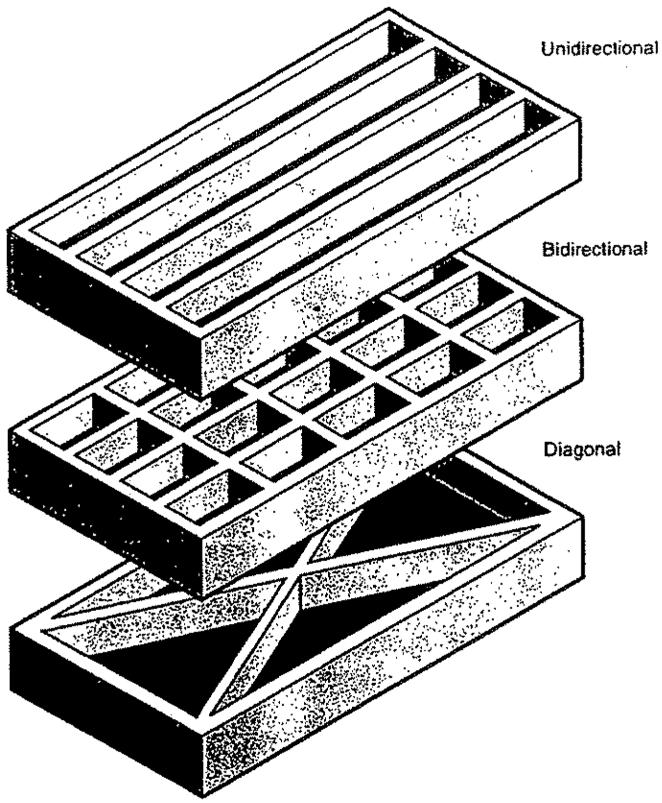


Figure 2-11



Different types of ribbing.

If your part needs torsional stiffness as well as longitudinal stiffness in both directions of the plane surface, use diagonal cross ribbing. Possibly the most-economical ribbing pattern for material usage, this ribbing configuration is easier to fill and less likely to trap air.

DRAFT

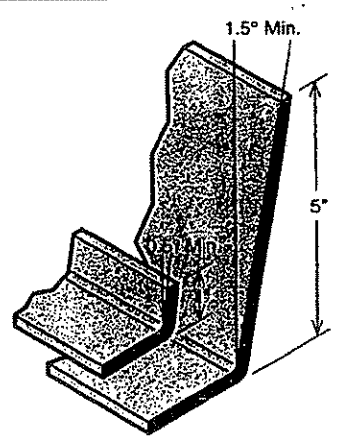
Every surface parallel to the direction of draw needs a draft angle to facilitate demolding. The recommended draft angle increases with part height.

Do not design deep wood grain textures on side walls. Even light textures in this orientation require additional draft. Elastomeric systems may require less draft in textured parts.

Generally, draft is more important on the core side (usually the top half of the mold) than it is on the cavity side, because parts generally shrink onto the core during cooling. Other rules of thumb for draft angles include:

- A minimum of 1/2° is usually adequate for parts with low side walls or ribs, typically those up to 1 inch deep.
- Add at least 1/4° of draft for every additional inch of draw, such that a 5-inch draw would require a minimum of 1-1/2° draft (see figure 2-12).

Figure 2-12 Recommended Draft



Rule-of-thumb for draft angle.

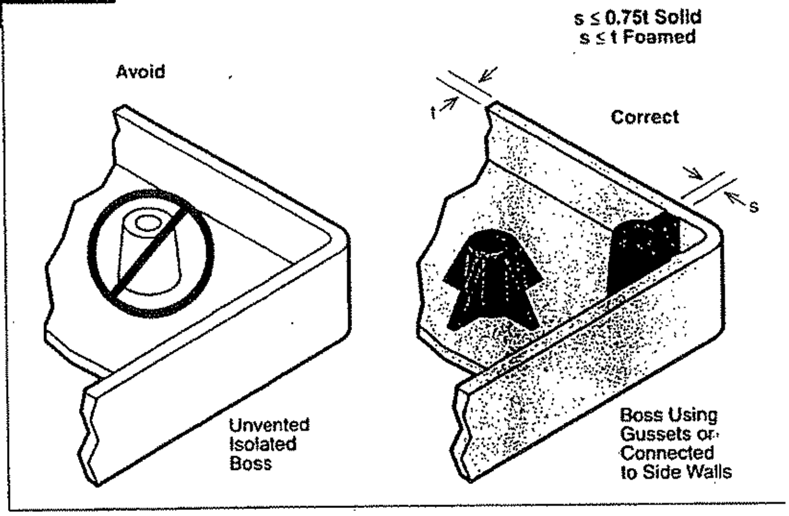
BOSSES

Use bosses for support, as spacers, or as attachment points. Attach bosses and other projections on the inside of parts to the side walls with connectors that allow air to escape during molding (see figure 2-13). Avoid isolated bosses, also known as "blind bosses." If you cannot attach a boss to a side wall because of interference or distance from the wall, design gussets or vent the boss with a core. Open bosses, those cored from one side and attached to an exterior side wall, are frequently used for assembly to eliminate the need for connectors or gussets (see figure 2-14). All bosses should have radii at their bases. Follow standard radii recommendations listed under the sections on solid, foamed, and composite materials in this brochure.

If you are using a boss to accommodate an insert, such as a screw or press fit, make the hole as deep as possible, preferably leaving only one nominal wall thickness to prevent sink marks. Other design guidelines include:

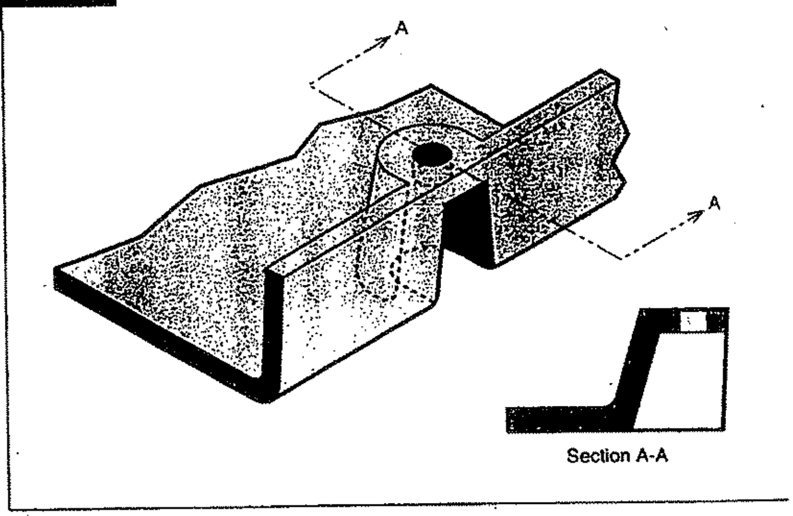
- If you cannot avoid an isolated boss, add gussets that extend from the base to the top on the side in the direction of flow to facilitate air removal and mold filling.
- Attach bosses to side walls with a connector of nominal wall thickness for foamed materials and 3/4 nominal wall thickness for solid materials.

Figure 2-13



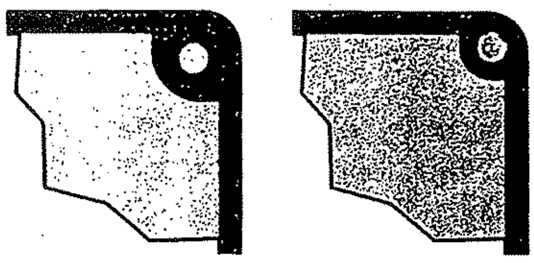
Bosses and venting.

Figure 2-14



Open boss.

Corner Bosses **Figure 2-15**



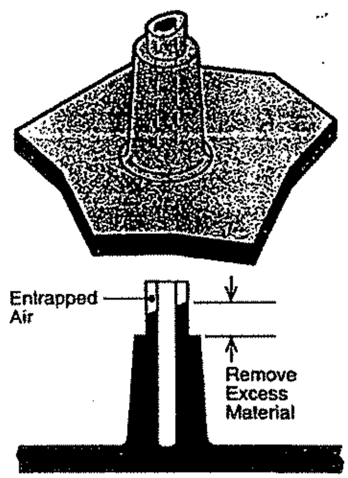
- Design bosses away from corners unless the boss can be connected to the wall directly (see figure 2-15) or indirectly (see figure 2-13). This will help prevent localized heat build up and possible warpage.

- Consider molding a hollow boss to maintain nominal wall thickness (see figure 2-16).

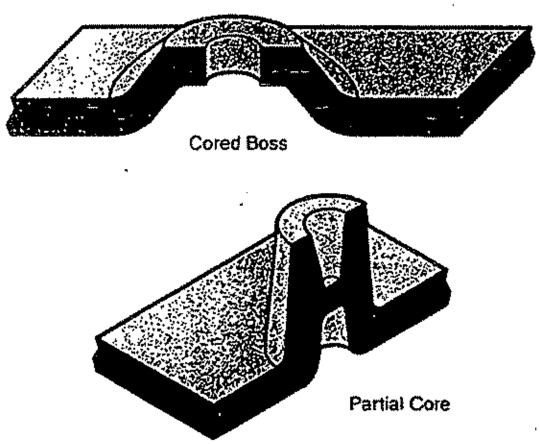
- Core bosses instead of drilling when using thread-cutting screws and thread-cutting inserts in parts made of structural foam to increase pullout strength.

- Consider designing an elongated boss and having the excess ground off as a postmolding operation, only as a last resort (see figure 2-17).

Elongated Boss **Figure 2-17**



Hollow Bosses **Figure 2-16**

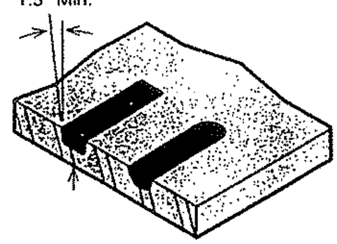


HOLES, GROOVES, AND SLOTS

Holes can be postdrilled, molded in the direction of draw, or formed by a retractable pin actuated by a hydraulic cylinder. A hole in a side wall with enough draft can also be formed by having the mold core and cavity meet at the hole (see figure 2-18). In this design, holes can be positioned anywhere on the wall.

Orient grooves and slots in the flow direction to minimize air entrapments or knit lines. Make sure that grooves are rounded or chamfered rather than sharp to help flow, vent air, and reduce stress concentrations (see figure 2-19).

Grooves **Figure 2-19**



Grooves should not reduce the wall thickness to the extent that foam flow is impeded. As a rule of thumb, do not recess grooves more than 3/16 inch for foamed materials and 3/32 inch for solid materials. Groove width should follow the rules established for slots in figure 2-22. Wider grooves run the risk of racetracking and air entrapments.

Consider positioning slots in a side wall, curled around the base plane, to allow for molding without slides (see figure 2-20). Another option is to design slots with stepped cutouts, positioned in a sloping wall section (see figure 2-21). Thicker walls will require more slope. If using this last option, do not make the mating sections too sharp, as this could damage the mold. Rules of thumb

Hole in Side Wall **Figure 2-18**

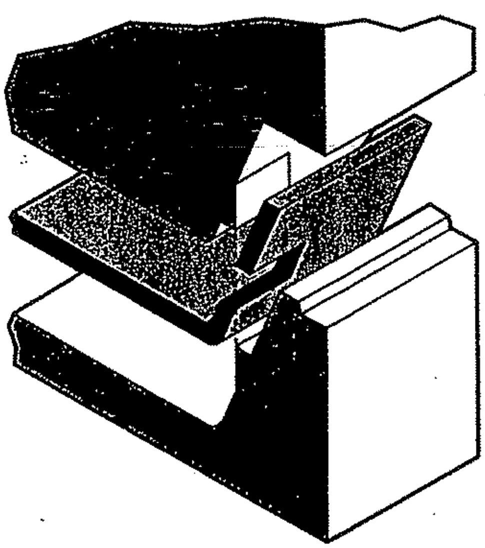
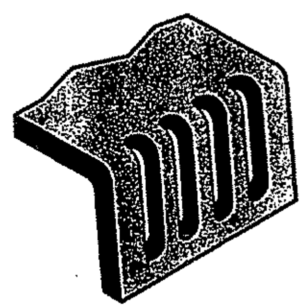


Figure 2-20



Slots curled around corner.

Slots or Louvers on Sloping Wall

Figure 2-21

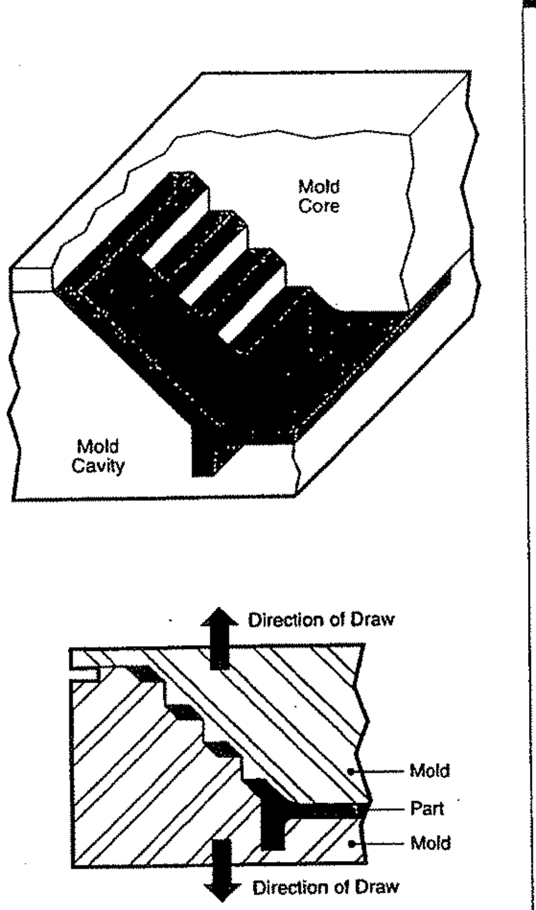
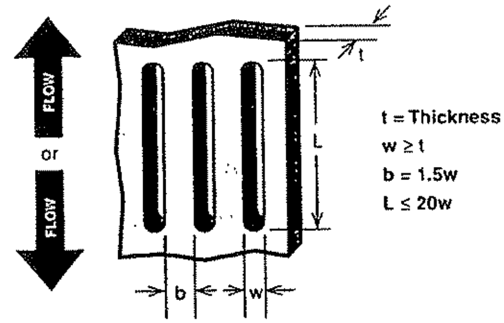
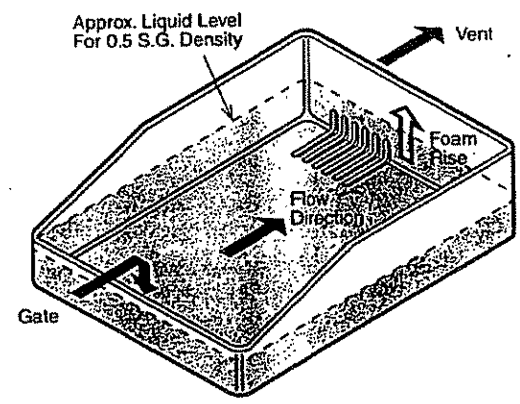


Figure 2-22



Basic dimensions for slots.

Figure 2-23



As a good design practice, locate slots, grooves, and holes under liquid level. Place in foam-rise direction.

for slots or louvers are shown in figure 2-22.

Design slots and grooves with a minimum 1-1/2° draft to help with demolding.

In foamed materials, grooves, slots, and holes should be located under the liquid level and lie in the direction of foam rise to help prevent air entrapment (see figure 2-23).

INSERTS

Polyurethanes have low molding temperatures and pressures, making them ideal for encapsulating reinforcing inserts. The insert should not impede material flow. If using a hollow insert, the ends must be sealed. Thermoplastic end caps have been successfully used to seal inserts. To promote good adhesion with the polyurethane, clean and roughen the inserts and, if necessary, treat them with an adhesion promoter.

The type of RIM system used determines the recommended minimum distance between an insert and the mold wall. For solid materials, this minimum distance is 1/8 inch; for foamed systems, 1/4 inch. For example, a solid material with a 1/8-inch nominal wall thickness should have a minimum distance of 1/8 inch between the wall and insert (see figure 2-24).

Encapsulated inserts are used for any number of reasons. For example, they:

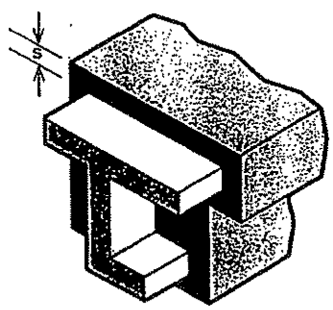
- Increase stiffness
- Reduce wall thickness
- Absorb high stresses
- Control thermal expansion

The most-common types of inserts are discussed in the following sections.

Metal Stiffening Inserts

Molding metal inserts into RIM polyurethane materials will increase stiffness significantly. Inserts of all types — including flat plates, extrusions, tubes, and bars — have been easily and successfully encapsulated. Fully encapsulating inserts eliminates metal corrosion, while reducing thick cross sections, controlling deflection and thermal elongation, and absorbing high stresses.

Figure 2-24

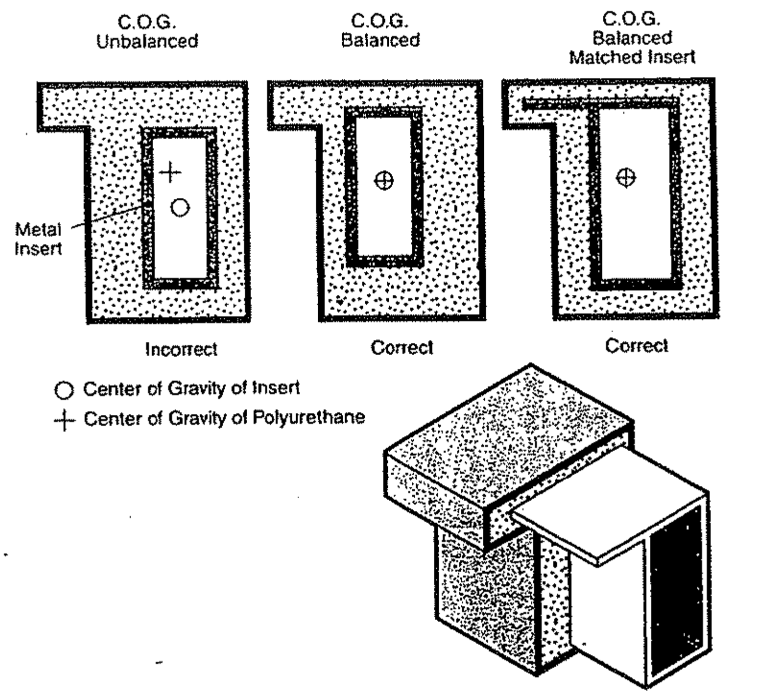


s = 1/4" for Foamed Systems
s = 1/8" for Solid Systems

Minimum wall thickness for using inserts.

Calculate the centers of gravity for both the RIM material and metal insert to reduce the potential for warping (see figure 2-25). The centers of gravity should coincide to prevent the part from bending because of the movement due to the differences in the coefficients of linear thermal expansion. As the temperature increases, the polyurethane material will be in compression and the metal insert in tension. As the temperature decreases from the ambient, the reverse is true: the polyurethane material is in tension; the insert in compression. The relative cross-sectional areas of the two materials determine the ultimate elongation of the part.

Figure 2-25



Balancing the cross-sectional centers of gravity.

When using Baydur structural foam, molded-in inserts may offer greater pullout strength, because skin forms over the entire insert surface. When using press-fit inserts with structural foam parts, mold the hole so that skin forms inside.

Generally, molders prefer press-fit inserts, even though these inserts may not be as strong as molded-in ones. Placing inserts on pins inside the mold can increase cycle time significantly. Although rare, inserts may also fall off pins during molding.

The insert design, hole diameter, part density, and screw size determine the pullout force and stripping torque of threaded inserts. Contact your insert manufacturer for more information about obtaining whole-diameter values. See Bayer's *Plastics: Joining Techniques* manual for more information on inserts.

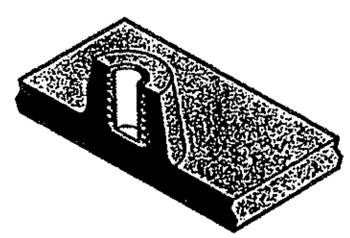
Wood Stiffening Inserts

Wood inserts — generally less expensive and lighter than metal inserts — can also be used as stiffening inserts in polyurethane parts. When a finished part is subjected to repeated loads, wood inserts may separate from molded polyurethane if the wood's moisture content exceeds 6%. If the wood insert cannot be dried to meet this limit, it must be sealed with a lacquer before molding.

Threaded Inserts

Threaded inserts are particularly useful when components must be attached to RIM-molded parts (see figure 2-26). Use appropriately sized, press-fit inserts with respect to boss-hole diameter. Use threaded inserts if your part is going to be frequently assembled and disassembled.

Figure 2-26

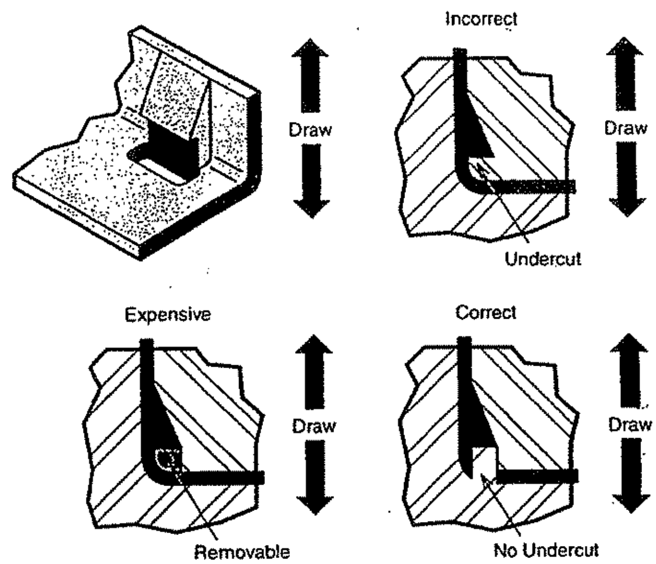


Typical threaded insert.

UNDERCUTS

If possible, avoid undercuts when designing parts made of rigid RIM polyurethanes (see figure 2-27). They add to cost and may create demolding problems. Modify the part geometry or mold orientation, or divide your part into two separate molds to avoid undercuts. For parts made of elastomeric materials — including reinforced RIM Bayflex systems — minor undercuts can be a design advantage. The flexible nature of these materials accommodates easy mold release even with minor undercuts.

Figure 2-27

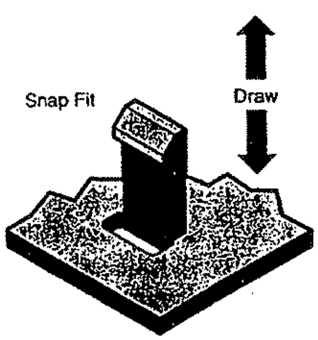


Mold configurations showing undercuts.

SNAP FITS, WIRE GUIDES, AND HINGES

A simple, economical, and rapid joining method, snap-fit joints offer a wide range of design possibilities. All snap fits have a protruding part on one component — a hook, stud, or bead — which deflects briefly during joining and catches in a recess in the mating component, thus relieving the deflection force (see figure 2-28). For more information on snap fits, please ask for a copy of Bayer's *Snap-Fit Joints* brochure.

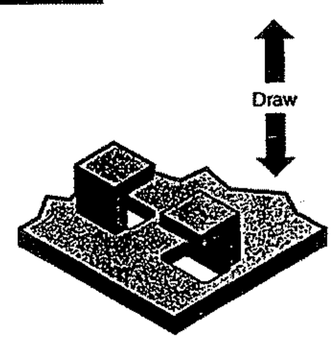
Figure 2-28



Snap-fit hook molded through hole to form undercut.

Used extensively in business-machine and appliance housings, wire guides offer simple design solutions to keep cables in position (see figure 2-29). Generally molded into the part, wire guides can be designed as a restraint that is molded without undercuts.

Figure 2-29



Wire guides.

When designing hinges, consider the end use: will it be a permanent connection, will it be used often, and/or will it have to disengage after a certain opening angle. All of these factors will affect design.

For permanent, frequently used joints, consider metal hinges, which can be molded-in or postmold assembled. While they add to costs, they may be optimum in long-term applications.

For permanent, infrequently used hinges, consider the living hinge (see figure 2-30). Typically, they are made of the same material as the part, but can be made of a different material. Bayflex elastomeric materials have excellent flexural fatigue strength. Molded strips of Bayflex elastomers can be cut and placed into a mold to form a living hinge for a more rigid part. However, if such a hinge breaks, it will be virtually impossible to repair.

Another hinging method is to mold a part that looks and operates like a metal hinge, with alternating sections on opposite part halves. These partial hinges offer a designer a method of forming hinges without undercuts (see figure 2-31). While they have a reduced load-carrying capability, partial hinges offer lower tooling costs and use hinge pins, as full metal hinges do. A rod pushed through the assembly completes the hinge. This design will disengage when the joint angle reaches 180°. If you do not want the hinge to disengage, consider designing full holes at the ends with a retractable core pin.

FILLERS

Using materials that have glass or other inert fillers will affect your part's shrinkage, coefficient of linear thermal expansion (CLTE), stiffness, and impact strength. A filled Bayflex elastomeric polyurethane material can have a CLTE closer to steel. Generally, fillers include fiberglass flakes, short glass fibers, or other mineral fillers. Usually, fillers need to have a sizing treatment to promote adhesion.

As filler content increases, stiffness increases. Short fibers usually orient in the direction of flow, causing greater

Figure 2-30 Living Hinge

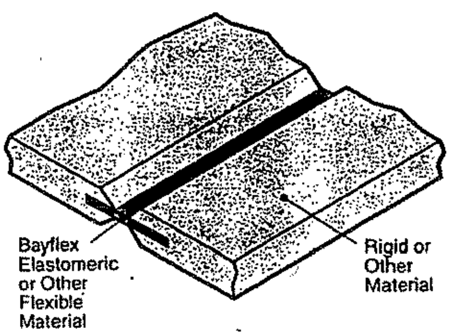
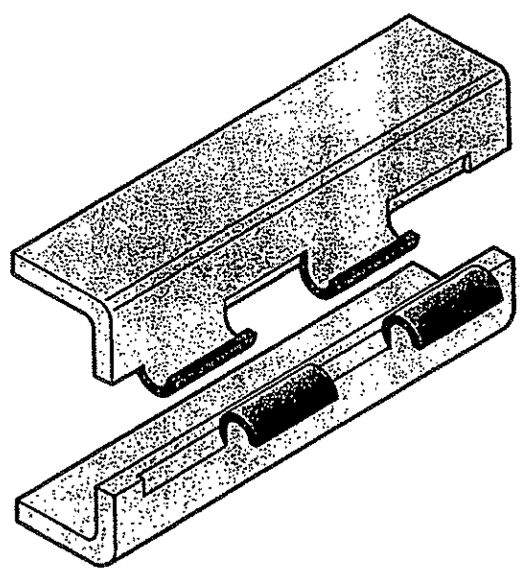


Figure 2-31 Partial Hinges



CW 0308707

note filled vs diff in CLTE

stiffness and lower CLTE parallel to the fiber orientation. Adding 15% glass filler to a Bayflex elastomer can almost double its flexural modulus. Test your part to ensure that it performs acceptably with the suggested filler content. When specifying materials with fillers, always check the material safety data sheet from your filler supplier for safe-handling practices for their products.

WARPAGE IN PART DESIGN

Warpage has many causes, including uneven mold and part cooling, incorrect positioning of inserts, unfavorable part geometry, and forces caused by incorrect stacking before a part has fully cured. As a designer, you should be aware of the potential for part warpage early in the design process.

Plastics have significantly higher CLTEs than metals, a major consideration if you are designing a part with structural metal inserts. Please refer to the section on metal stiffening inserts in this brochure for more information.

Warpage is more noticeable in flat parts than in those with more complex geometries. Table 2-1 lists typical coefficients of thermal expansion.

Please see the section on back molding on the next page for more information on how to help avoid warpage with dissimilar materials.

Table 2-1 Coefficients of Linear Thermal Expansion (CLTE) for Common Materials

Material	in/in/°Fx10^{-6}
Steel	6
Composite RIM	8
Nylon GF*	13
Polycarbonate GF*	17
Baydur XGT-GF	28
Polycarbonate	39
ABS	44
Nylon	45
Polyester	60
Baydur Structural Foam	65
Polyethylene	70
Elastomeric RIM Unfilled	78

*glass-filled resins

CREEP CONSIDERATIONS

All materials show a certain amount of irreversible deformation under long-term load, known as creep. Polymer-chain movement under stress causes creep in polyurethane materials. Creep is usually measured in tension or flexure, with measurements taken at several different temperatures and at different loads.

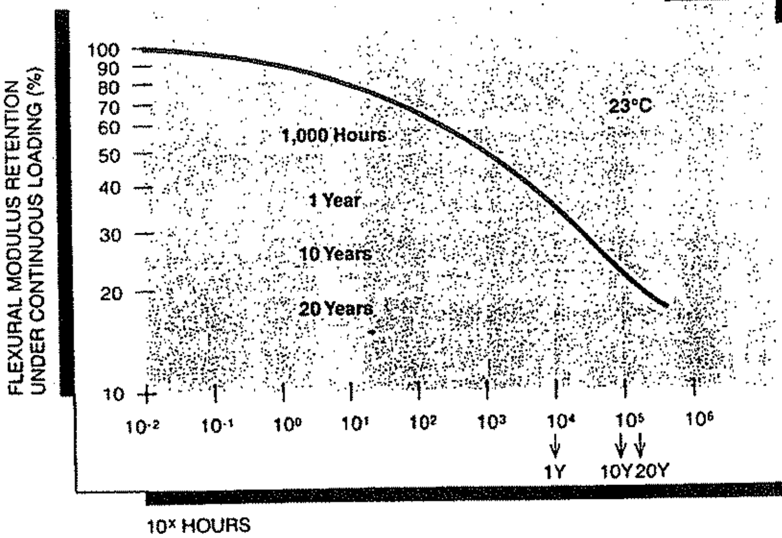
Figure 2-32 shows the data obtained in a flexural-creep test for a Baydur foamed system, normalized to show the retention in modulus over time. This data should not be taken out of context for two reasons: 1) The data represents parts subjected to continuous loading; 2) Manufacturers usually require instantaneous displacements to be very small. Manufacturers and designers should determine acceptable safety factors for the part's life.

FATIGUE CONSIDERATIONS

Repeated loading causes fatigue, a progressive, permanent change in a part subjected to cycling stresses and strains. While at first no noticeable damage may appear, over time and with continued stresses, parts may begin to fail. For instance, consider a discharge chute on a lawn mower. As you use the mower.

Modulus Retention for Baydur Structural Foam

Figure 2-32



the chute occasionally bumps into a tree or wall. While at first there is no apparent damage, after several years, you will notice that the chute has cracked.

Typically, fatigue tests consist of repeatedly putting a sample under tension. Generated results show a material's ability to endure these repeated loads.

BACK MOLDING

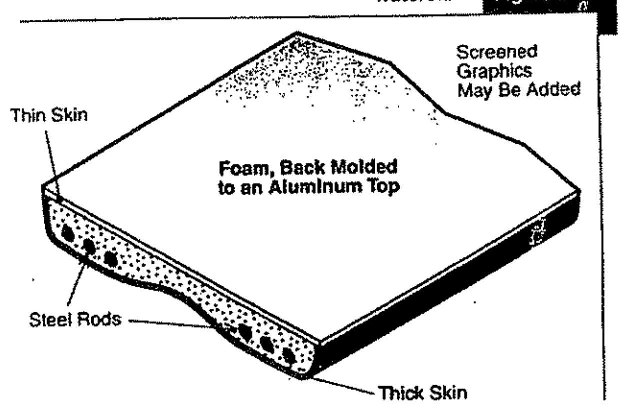
One unique design feature of RIM systems is that other material types, such as vinyl, metal, glass, polycarbonate, acrylic, and others can be placed in the mold prior to molding. Polyurethane materials will then mold against this second material. When using RIM systems for back molding, one-sided molding against a different material, some warping can result, even when the CLTEs are similar. While most methods to address warpage are application-specific, consider these general suggestions:

- Place additional inserts to balance the centers of gravity (see figure 2-25).
- Adjust the cooling system in the molding process.
- Modify design of the substrate.
- Create specialized jigs for postcuring.

For instance, because of its different rate of heat conduction, a metal sheet — such as those used in the production of snow skis — may cause different skin thicknesses on the opposite sides of the part made of structural foam (see figure 2-33). Different skin thicknesses exhibit different shrinkage behavior, possibly leading to warping. Inserting steel rods just under the skin on the side opposite of the sheet metal can help compensate for this warpage. Contact your Bayer representative for more information to address this topic.

Waterski

Figure 2-33



Filled Bayflex and rigid PRISM systems can have high flexural moduli, making them a good choice for thin-walled applications. Because of their excellent impact properties, flexibility, toughness, and ductility, elastomeric solid materials find many uses in automotive panels and bumper fascias. They also have excellent resistance to scratching and tearing. Rigid, solid materials are good choices for business machines, electronic and medical housings, load-bearing applications, appliances, and consumer-product housings. Typically, parts made of solid polyurethane materials incorporate many of the same design principles as those made of thermoplastic resins.

WALL THICKNESS

Parts made of solid polyurethane materials have similar wall thicknesses to those made of thermoplastic materials (1/16 to 1/4 inch). Additionally, RIM parts can have walls as thick as 1-1/2 inches. A wall thickness of 1/8 inch for solid PRISM systems, or 1/4 inch for Baydur GS systems, is typical for parts that need a UL 94 V0 and 5V rating. Please note that flammability results are based upon small-scale laboratory tests for comparison purposes only and do not necessarily represent the hazard presented by this or any other material under actual fire conditions.

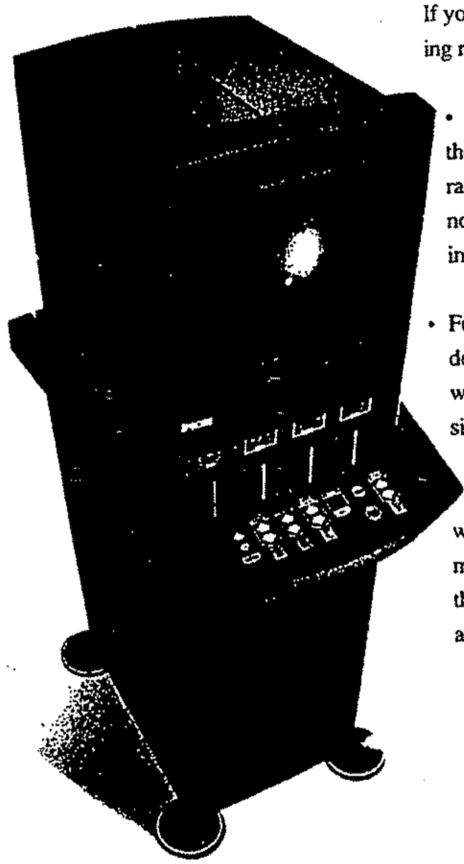
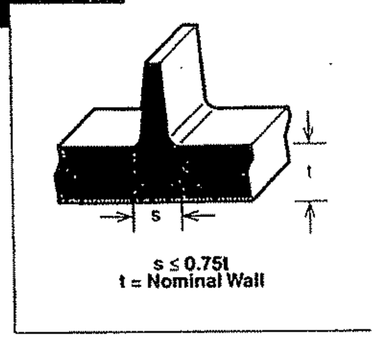


Figure 3-1



Rib/wall ratio for solids.

RIB DESIGN AND CONFIGURATION

If your part requires ribs, use the following rules of thumb with solid systems:

- For solid materials the thickness at the rib root — including both sides of radii — should not exceed 75% of the nominal wall thickness for parts requiring a show surface (see figure 3-1).
- For non-aesthetic applications, you can design thicker ribs, up to the nominal wall thickness. Your part may develop sink marks causing visual blemishes on the surface opposite the rib (see figure 2-7). These sink marks appear where the protrusion and mating wall meet because the locally increased wall thickness leads to increased shrinking as the part cools.

RADII/FILLETS

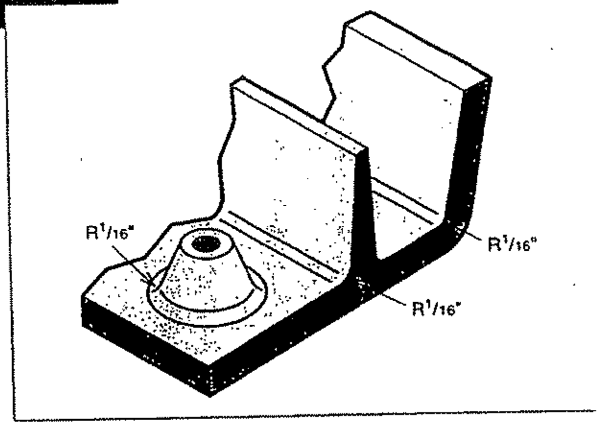
Radius the inner corners of all ribs, bosses, and walls at least 1/16 inch to reduce stress concentrations and help avoid air entrapment (see figure 3-2). Outside corners are not as susceptible to stresses and may not need radii.

BOSSES

When designing bosses in parts made of solid polyurethane materials, allow a wall thickness equal to 75% of the part's nominal wall thickness around the cored hole (see figure 3-3). Hole depth should allow for a maximum of one nominal wall thickness of material at the bottom. Follow rib-design suggestions for radii and draft angles.

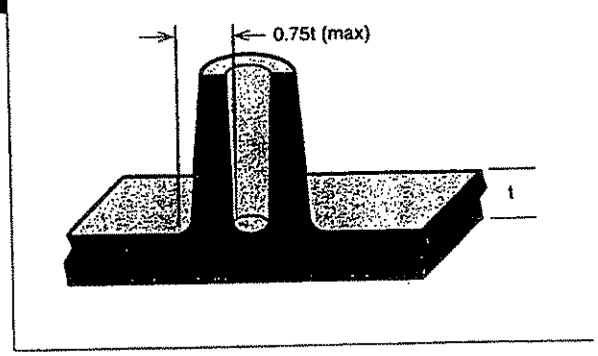
For the diameter of the boss hole, refer to the specifications of the insert or screw manufacturer. For example, inserts greater than one inch in diameter have been successfully used in the field.

Figure 3-2



Correct radii/fillets for solids.

Figure 3-3



Boss dimensions.

Baydur structural foamed systems offer excellent strength-to-weight characteristics, because of their sandwich-like structure. These polyurethane materials are found in many large parts, such as electronic and business-machine housings, cab roofs, consoles, cabinets, and shelves. Additionally, structural foamed materials are used extensively in aquatic sports equipment, such as skis and knee boards, because their density is lower than water, which allows them to float. They also offer designers more latitude than other materials: wall thickness can be varied; sink marks from ribs and bosses are less common.

While Bayflex systems are not used in structural applications and are not subject to the same design restrictions, they offer some unique capabilities for designers. For instance, because Bayflex polyurethane materials have high compressibility, you can design small undercuts without slides. For more information on design parameters, contact your Bayer representative.

FOAM RISE AND FLOW

The liquids used in foamed systems fill a percentage of the cavity's volume, depending upon the final part density required. The level of liquid placed in the mold prior to expansion is referred to as the "liquid level." The remaining volume is filled as the liquids react, creating foam. Foam expands upward and outward, a process called **foam rise**. Foam density can range from 0.3 to 1.0 g/cm^3. Most parts have a foam density between 0.6 and 0.8 g/cm^3.

While freon was used as a blowing agent for many years, all Bayer's RIM polyurethane systems are CFC- and HCFC-free today. Growing environmental concerns, coupled with advances in water-blown systems, have led to alternatives.

WALL THICKNESS

Wall thickness plays an important role in designing parts made of foamed materials. Baydur structural foam has been used in parts with localized wall thicknesses as thin as 1/8 inch to as thick as over 1 inch, although the typical nominal wall thickness ranges from 1/4 to 1/2 inch. Other rules of thumb include:

- Increase the wall thickness and reduce the part density to maintain part weight and optimize part stiffness.

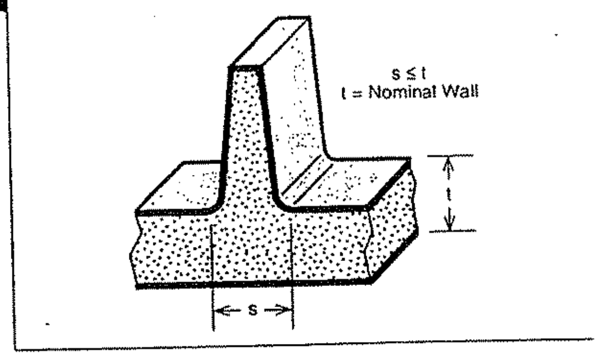
- Avoid having wall-thickness increases at the end of foam rise, near the vents. The increasing viscosity and resulting drop in flowability can lead to air entrapment, bad knitlines, and insufficient packing.

- Keep the ratio of wall-thickness change below a factor of two, if possible. While some award-winning applications have deviated from this rule of thumb, the differential friction in the cross sections can cause racetracking, leading to venting problems and surface imperfections (see figure 2-4).

- Keep the larger thickness changes under the liquid level.

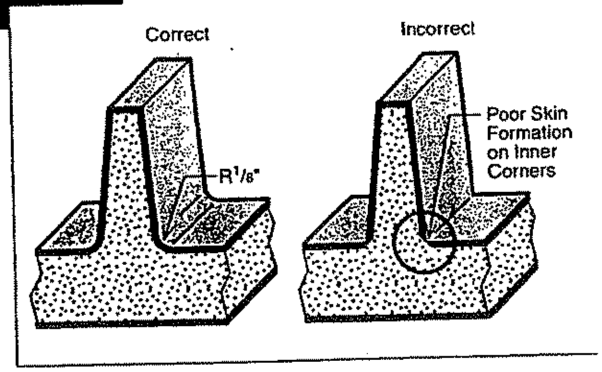
- Avoid excessively thick cross sections as they cause long demolding times. Consider using a space-filling insert to help fill thick cross sections.

Figure 4-1



Rib/wall ratio for foamed systems.

Figure 4-2



Effect of radius on skin formation.

RIB DESIGN AND CONFIGURATION

When designing ribs for parts made of foamed materials, the rib-root thickness including both sides of radii should not be greater than 100% of the nominal wall thickness to help avoid sink marks (see figure 4-1). When aesthetics are not a primary concern, you can design ribs thicker than the nominal wall thickness. Other considerations include:

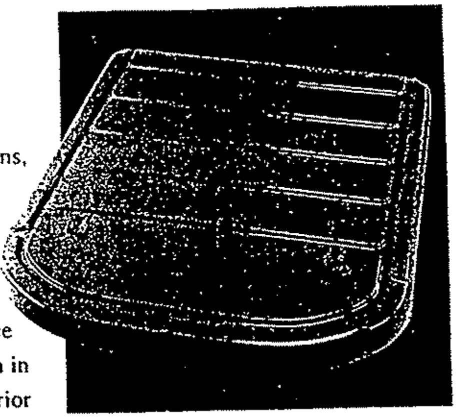
- Locate ribs in the direction of foam rise and flow. If this is not possible, provide for venting to prevent air entrapment.

For more information on ribbing direction and configurations, please refer to the general design considerations in this manual.

RADII/FILLETS

Radius the inner corners of all fillets and adjacent walls a minimum of 1/8 inch to reduce stress concentrations, promote good skin formation, and help avoid air entrapment (see figure 4-2). Inside corners on parts are more difficult to cool than large, flat areas. This temperature difference can lead to poor or no skin formation in sharp inner corners, resulting in inferior mechanical properties. Outside corners are not as susceptible to poor skin formation and will form thick skins. Other rules of thumb include:

- Do not make sharp transitions. Radius corners 1/8 inch to allow for proper skin formation.
- Radii are extremely important for parts made of foamed rigid systems because they are generally more notch sensitive than parts made of Bayflex solid systems.



Roof for cab of John Deere combine measures 69 inches by 62 inches and weighs 50 pounds.

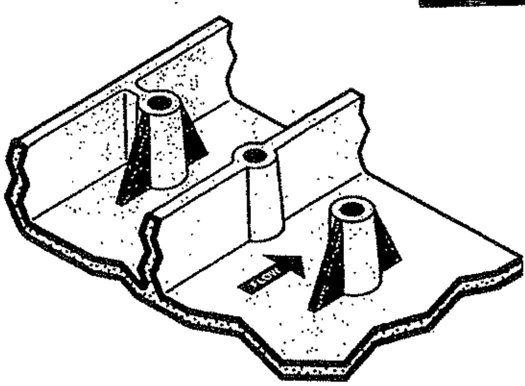
BOSSES

Bosses facilitate mechanical assembly or act as supports or spacers. Connect them to the part's outer walls or design them with gussets (see figure 4-3). When bosses are used to accommodate screws or inserts, consider the following suggestions:

- Make the minimum wall thickness 3/16 inch around the hole if an insert is used.
- Design bosses, including radii, with a wall thickness no greater than the nominal wall thickness around the cored hole (see figure 4-4).
- Core bosses when using self-tapping screws and inserts to form skin in the hole (see figure 4-5).
- Radius all bosses 1/8 inch at their bases.

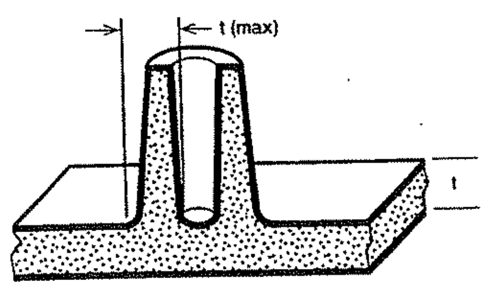
For the diameter of the boss hole, refer to the specifications of the insert or screw manufacturer. For example, inserts greater than one inch in diameter have been successfully used in the field.

Figure 4-3



Boss versus flow direction.

Figure 4-4



Boss dimensions.

STRUCTURAL ANALYSIS CONSIDERATIONS

Because the mechanical properties are not uniform throughout a typical cross section in a non-isotropic material, the available physical-property data may not allow you to calculate reasonable deflections. Combining properties obtained from different testing methods is required for parts made of structural foam. For example, flexural modulus helps to predict deflection on horizontal surfaces (perpendicular to the load direction) and tensile modulus for vertical surfaces (parallel to load direction). Parts with complex geometries may require using both flexural and tensile moduli to help predict real-world behavior.

Finite-element analysis can use these values to estimate the part's displacement field. This type of analysis will complement prototype testing, but should not be substituted for prototype testing under actual, end-use conditions. For more information on analyzing Baydur structural foam, request a copy of *The Performance of RIM Structural Foam in Load-Bearing Applications* from Bayer.

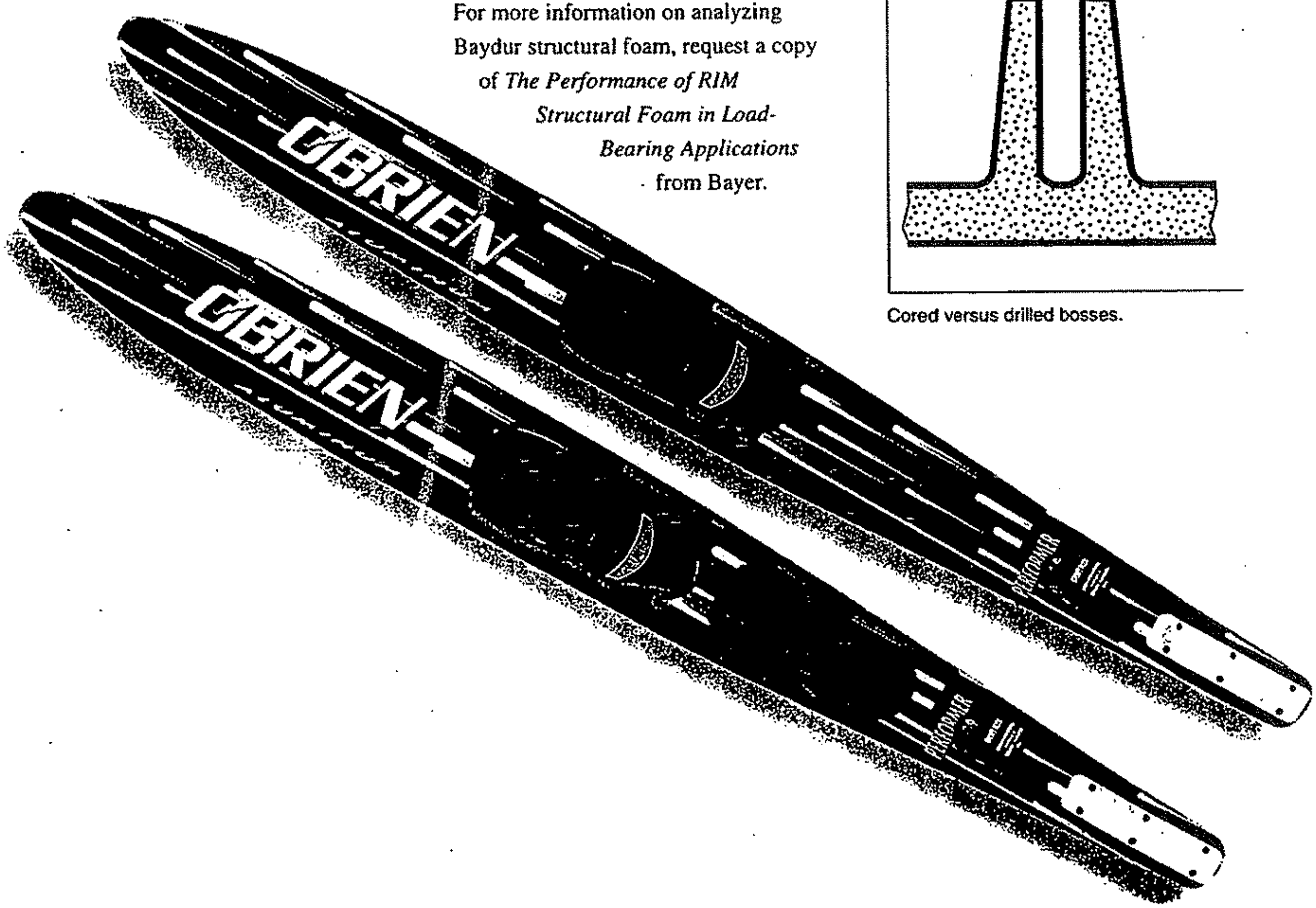
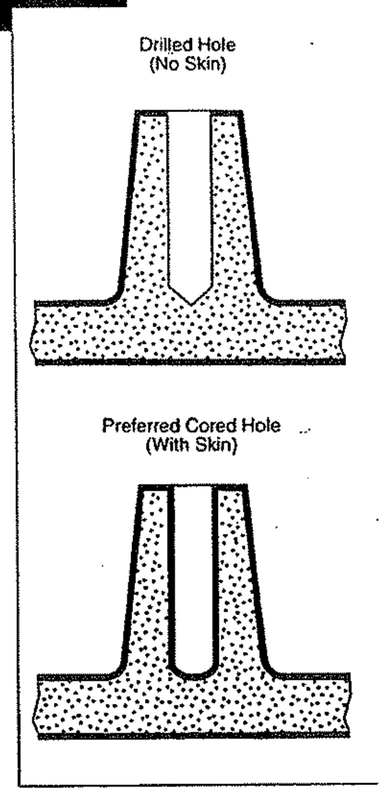


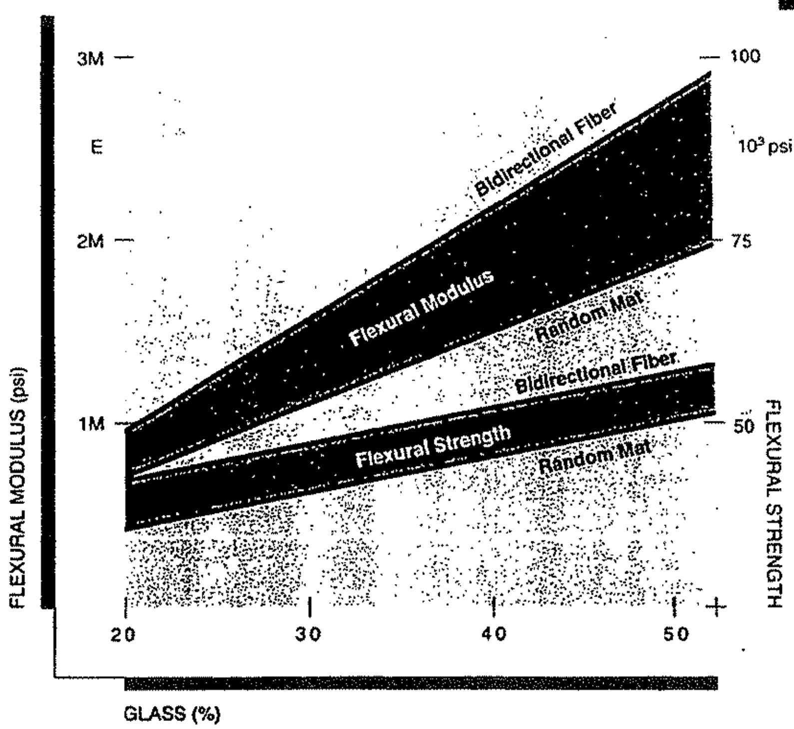
Figure 4-5



Cored versus drilled bosses.

Composite materials offer a unique benefit to part designers: you can place localized additions of glass mat within the mold to strengthen higher-stressed areas. This flexibility allows for enhanced material properties. Load-bearing parts made of Baydur structural RIM materials offer excellent impact strength and high flexural modulus.

Flexural Properties Versus Percent/Type of Glass for Baydur STR/C **Figure 5-1**



GLASS MAT

Composite materials, made of a polyurethane system reinforced with glass-mat fiber, have an extremely high flexural modulus that depends upon the fiberglass content (measured by weight), its location, and the direction of the mat fibers. As the percentage of glass increases, the flexural modulus of the part increases. The highest modulus attained thus far is approximately 3.0 million psi in a part made of Baydur STR/C composite with 60% glass in several layers of bidirectional mat. At 30% glass, a flexural modulus of 1.2 million psi with random-fiber mat and 1.5 million psi with bidirectional mat are not uncommon. The flexural modulus and part stiffness are greater in the direction of the glass fiber.

Glass mats are available in random or directional configurations, with various-diameter fibers. Glass mats are fitted in the mold prior to injection. Resistance to flow increases as the percentage of glass mat increases. This resistance to flow, causing back pressure, can make mold filling difficult. Always check the material safety data sheets (MSDS) and consult with your glass-mat supplier for safe-handling recommendations for their products.

If your part needs to be stiff and lightweight, consider using a foamed composite with more of a sandwich-like construction. This type of composite features glass fibers close to the surface, creating a less-dense cross section.

When the processor is using composite materials, he should follow these rules of thumb:

- Avoid specifying more than 50% glass in any area. At this high percentage, bulky glass can be difficult to compress in the mold and may be difficult to fill.
- Extend the mat to the mold edge. Undersizing mats may lead to low-glass areas — called resin-rich areas — around the periphery and create a preferential path for material flow. These resin-rich areas are weaker under load and exhibit more-brittle behavior than sections reinforced with glass mat.
- Use a thicker wall or a higher percentage of glass to increase stiffness. Figure 5-1 shows modulus and flexural strength as a function of the percentage and type of glass.
- Make sure that localized glass additions can be "wetted out" — thoroughly saturated with liquid polyurethane material — to minimize dry and unfilled areas.
- Position gates directly into the highest percentage of glass whenever possible. The mixture should flow from the higher-density glass to the lower-density glass to minimize dry areas.

REINFORCEMENTS

Do not design ribs in parts made of Baydur composite systems, as they are difficult to mold. Alternatively, consider designing in corrugations and/or box beams (see figure 5-2). To make a box beam, glass mat is placed around a low-density, preformed core or space-filling insert (see figure 5-3). The structural RIM system is molded around it to produce a continuous integral beam. The size of the cross-sectional area of the beam, along with the wall thickness and percentage of glass will determine the overall stiffness of the box.

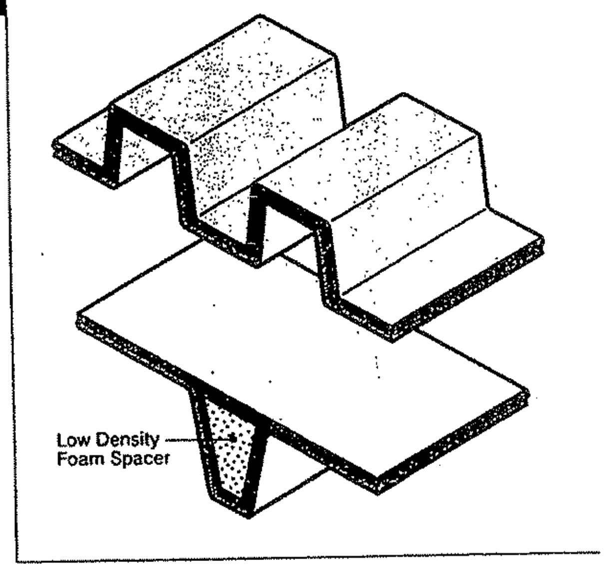
RADII/FILLETS

When designing corners, allow for an inner radius or fillet of 1/8 inch minimum to allow for continuous glass transition (see figure 5-4).

PADS

When working with composite materials, use pads or hollow bosses for assembly. Chamfer the ends of the pads for better mold filling (see figure 5-5).

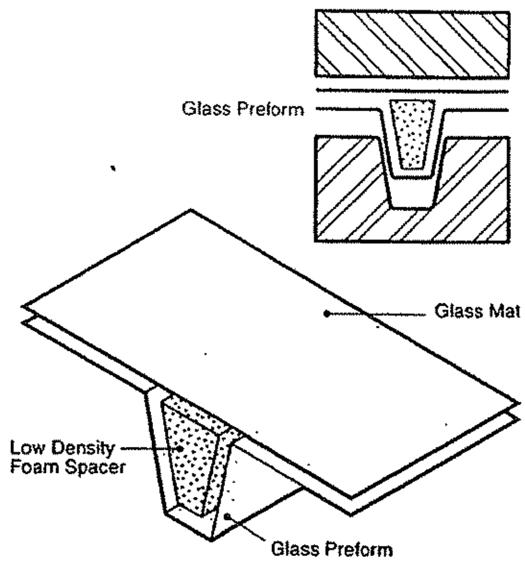
Figure 5-2



Corrugations and box beams.

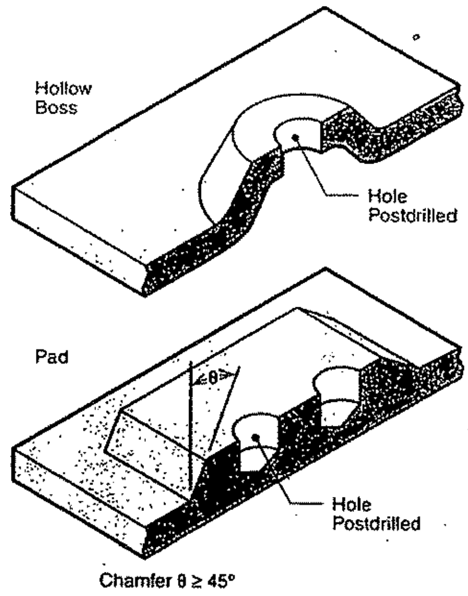
CW 0308717

Figure 5-3



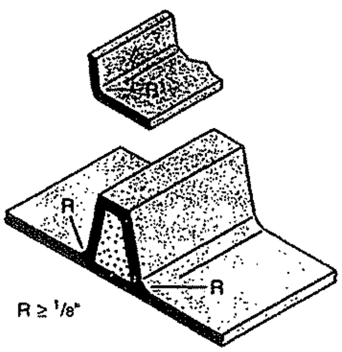
Continuous integral beam assembly.

Figure 5-5



Hollow bosses and pads for mounting.

Figure 5-4



Radii/fillet configuration.

PREFORMS

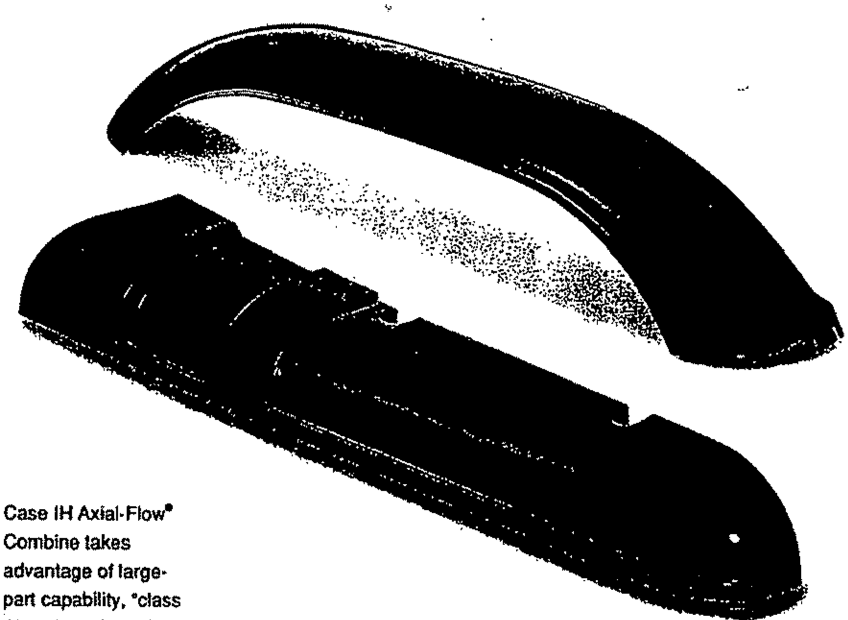
Preforms are required for complex parts with geometries that cannot be molded with a standard glass mat. They offer increased design flexibility for structural RIM parts, particularly in large production runs. Typically preforms are made of glass fibers that are held together by a thermoplastic or thermosetting binder in one of two ways:

- **Compression-molded preforms** use a glass mat treated with a binder. The mat is compressed into the desired shape in a heated mold. As the binder cools or cures, it glues the fibers into shape.
- **Spray-up preforms** involve spraying chopped glass and the binder onto a perforated, positive form. A vacuum on the opposite side draws the chopped glass onto the form, creating a random, spray-up pattern. This type of preform can accommodate more complex shapes, but may be more costly.

FINISHES

Structural RIM materials can accommodate many surface finishes, from textures to "class A." For visible parts requiring a "class A" finish, such as in automotive applications, specify a polished mold and surface-veil material. A veil is a very thin mat made of thin fibers that will help keep the glass fibers from protruding through the molded part surface.

In-mold coating is possible with Baydur STR systems. To create a good bond between the coating and glass matrix, allow the material to mold against a partially cured coating. For more information on in-mold coating, please contact your Bayer representative. The next chapter on postmolding operations provides more information on finishing.



Case IH Axial-Flow® Combine takes advantage of large-part capability, "class A" surface, in-mold coating, and lower-cost tooling for these Reinforced RIM (RRIM) applications.

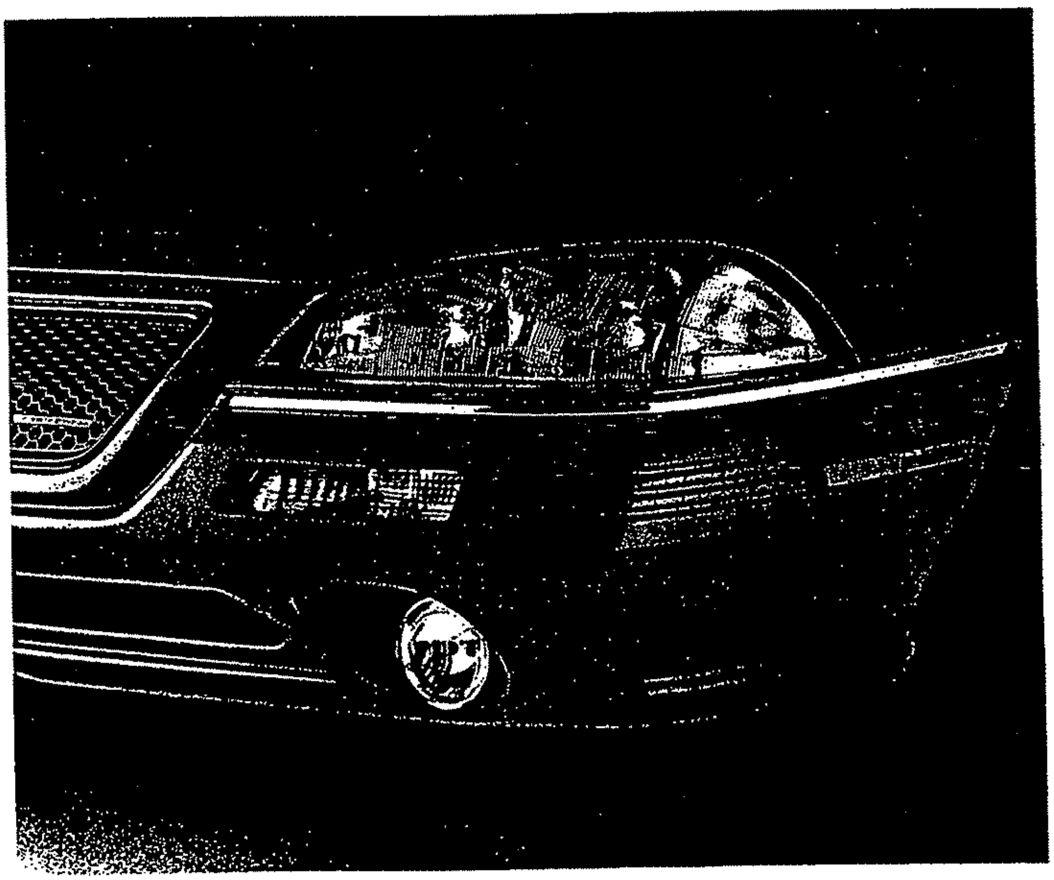
Most parts require postmolding operations, such as painting and assembling. These various operations are discussed in this section.

FINISHING

Parts made of polyurethane materials offer designers many options for color, texture, and other finishing considerations. For more information on any of these options, contact your Bayer representative.

Pigmentation

Pigments — organic dyes or color pastes added to the polyol — change the natural color of polyurethane materials. Polyurethanes are light to medium opaque brown when molded without color. Not inherently UV-stable, they will eventually yellow or gain a greenish tint when exposed to sunlight. While this discoloration does not affect physical properties, it is usually aesthetically objectionable. The amount of pigment added to the system is expressed as a weight percentage of the polyol, usually ranging from 3 to 10%, if your part will be painted.



Consider specifying pigments to give your final part a base color similar to the final surface coating. A higher pigment concentration is needed if your final part is going to be a dark color and you don't want to paint it. Most rigid RIM systems cannot be used to produce UV-stable, white or light-colored parts without painting. Pigments also will not hide surface imperfections and can cause color striations.

In-Mold Coatings

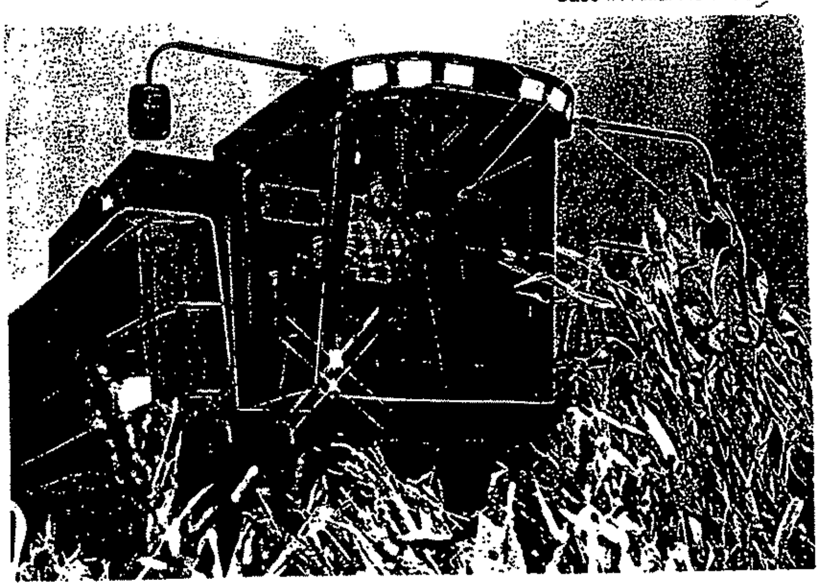
Consider in-mold coatings — special paints sprayed onto mold surfaces — as an alternative to postmold painting. After spraying, these paints dry for a brief period, so that the injected mixture flows over the semi-dry coating during mold filling. Typically in-mold painting is used for large, relatively simple molds, without complex details, such as agricultural-combine cab roofs and fenders. Other points to consider when selecting an in-mold coating include:

- In-mold coatings can save the cost of postmold painting.
- Mold surface cannot have imperfections, as every detail is reproduced.
- In-mold coating can reduce most secondary, finishing operations.

Patching

Occasionally, molded parts have air pockets, lower-density areas, and other small imperfections that may need to be repaired. If your part will ultimately be painted, a patching compound can fill these areas.

Case IH Axial-Flow® Combine



Smaller areas can use a single-component, commercially available patching compound; larger areas will need a two-component polyester compound, such as those used in autobody repair. For good patch adhesion, make sure the area is clean, free of mold-release agents or dust, and roughened. After the patch has cured, sand the area until smooth prior to painting.

Postmold Painting

While more costly than pigmentation, postmold painting offers the added benefit of exact color matching to other parts, and parts made of other materials. Postmold painting covers minor surface blemishes and allows similar parts to be painted in different colors. For example, a bumper made of polyurethane must match the steel side panels on the car to which it will be attached. However, the same bumper design is used on many different-colored cars. Postmold painting accommodates both of these design parameters.

Textures

Polyurethane molding techniques accommodate a number of different textures, including wood and leather grain, pebble, and graphics. For wood and leather grain or other fine textures, a nickel-shell mold can be used. This material is hard, has good release characteristics, reproduces textures well, and does not scratch easily. While molds accommodate custom-designed finishes with ease, pattern preparation can be expensive.

If considering a wood-grain finish, the mold must not have any nicks or scratches. Blemishes on the mold surface will appear in the finished part's grain, and wood-surface finishes cannot be retouched after molding. To complete a wood-grain effect, the grain is stained in a darker color, and the whole part is covered with a protective coat.

Pebble surfaces can be formed in the mold or added later with a coat of texture paint.

Graphics can be molded in. Simple masking will allow raised graphics and lettering to be painted in a contrasting color. For dense areas of text or small letters, consider using decals.

Decals and Silk-Screening

Decals work well with polyurethane parts, as long as the adhesion area has no texture and is clean and free of any release agents. While decals adhere well to painted parts, they will be somewhat easier to remove than those applied directly to unpainted parts.

Polyurethane parts can also be silk-screened. Contact your printer to discuss your needs.

ASSEMBLY OPERATIONS

Polyurethane parts that require assembly can be joined with screws, adhesive bonding, and nailing. This section gives an overview of some of these common joining methods.

Screws

One of the most cost-effective, reliable, and commonly used joining methods, self-tapping or wood screws can be used to assemble parts made of RIM polyurethane systems. To install screws, you can either drill or mold a hole in the part. If you chose to postdrill a hole, make it slightly smaller than the screw diameter, as you would with wood. Both methods yield relatively high pullout strength in parts made of Baydur structural foam.

- Generally, the pullout strength is proportional to the screw depth.
- Molded-in pilot holes will yield higher pullout strength than postdrilled holes for foamed materials.
- Parts made of structural foamed materials respond similarly to wood in most joining techniques.
- Parts made of elastomeric materials can be joined with screws, but the material can tear or stretch around the hole. Test your assembled part.

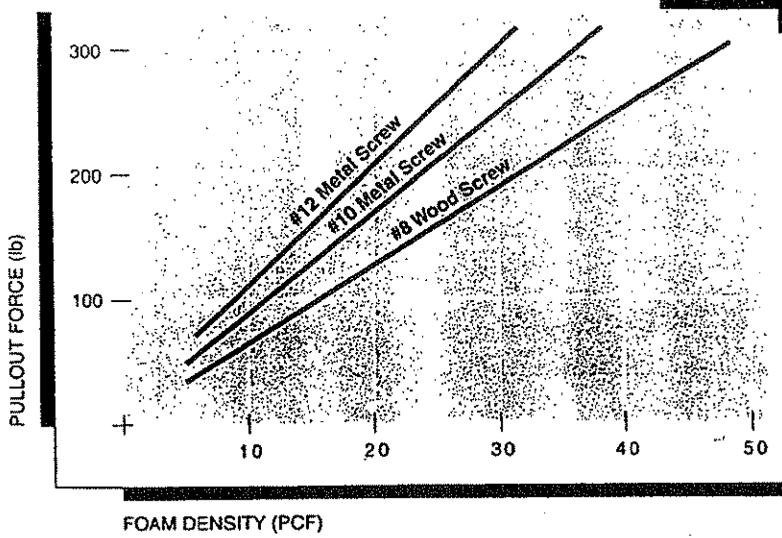
In foamed Baydur systems, parts with lower density make installing screws easier. The most-common foam density for screw installation ranges from 25 to 40 lb/ft^3 (0.4 to 0.65 g/cm^3), roughly comparable to that of wood. While several screws designed specifically for plastic are available, normal wood or sheet-metal screws can be used with foamed parts (see figure 6-1).

- For parts made of Baydur structural foam, use thread-cutting or thread-forming screws. Thread-forming screws may leave higher internal stresses close to the thread.

- For parts made of PRISM polyurethane, use thread-cutting screws.

Screw Pullout Strength Versus Foam Density

Figure 6-1



Screws can be inserted without a pilot hole into parts made of low-density foam. Use caution when installing screws, as the danger of stripping or foam breakout increases as the density decreases. Specify screws as a permanent attachment method on parts made of low-density foam. If your low-density part will be disassembled with any frequency, consider another joining method, such as threaded inserts. For more information on inserts, see the general part design section in this manual or request a copy of Bayer's *Plastics: Joining Techniques* design guide.

Adhesives

Polyurethane or epoxy adhesives work well with RIM polyurethane systems. The adhesion area in a lap joint should be at least three times the wall thickness. Bonds can have high strength in both tension and bending. Clean and roughen the adhesion areas to promote good bonding. For more information on adhesives, contact one of the following producers:

3M Industrial Specialties
St. Paul, MN 55144
612 733-1110

Ciba-Geigy Corporation
East Lansing, MI 48823
800 875-1363

Loctite Corporation
Newington, CT 06111
203 278-1280

Lord Corporation
Erie, PA 16541
814 868-3611

Ashland Chemicals
Columbus, OH 43216
614 889-3639

POSTFABRICATION

Nailing/Stapling/Planing

Many standard woodworking techniques can be used with Baydur structural foam, including sawing, drilling, nailing, stapling, sanding, and routing. If your part will be fabricated via one of these methods, design for common woodworking techniques. Avoid these techniques for mass-produced products as they are crude and labor-intensive.

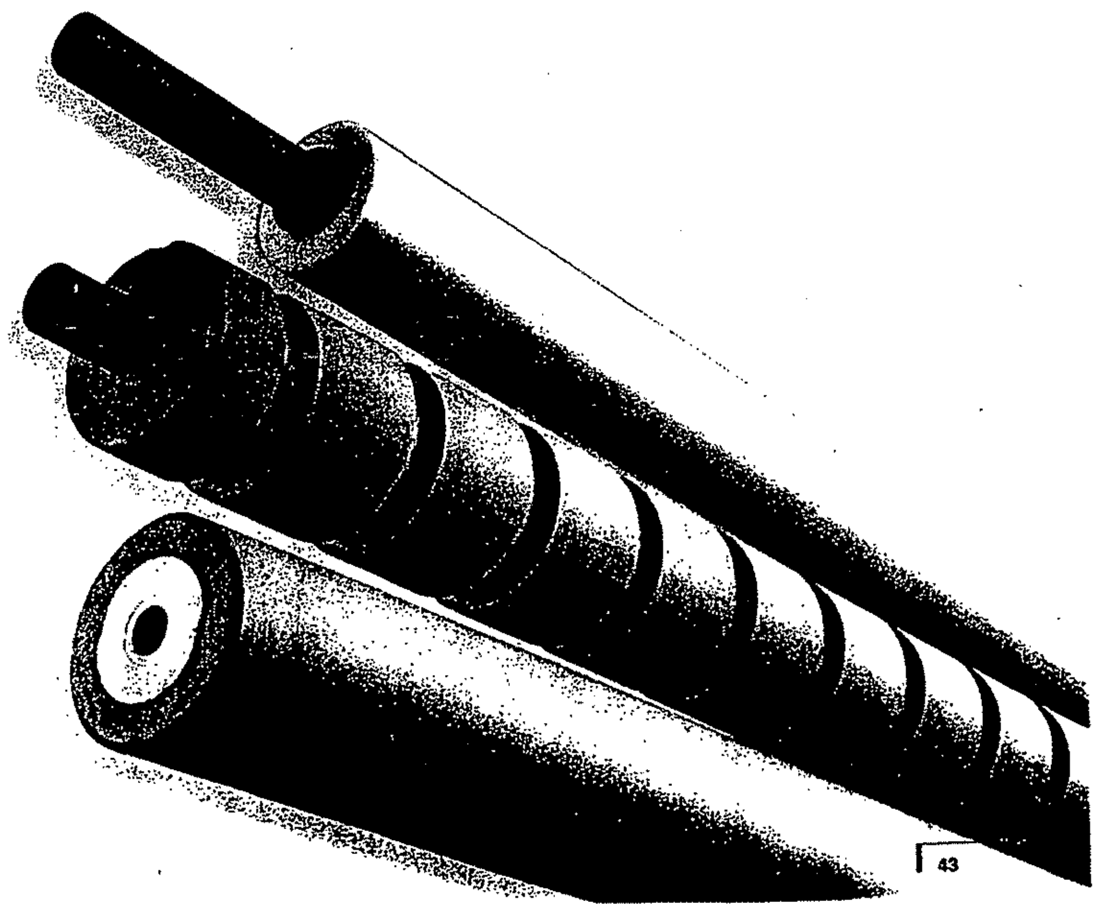
Do not plane foamed polyurethane parts. Planing will cause skin loss and possible exposure of foam core, with the resulting loss in structural integrity, as well as physical and mechanical properties.

Recycling Polyurethanes

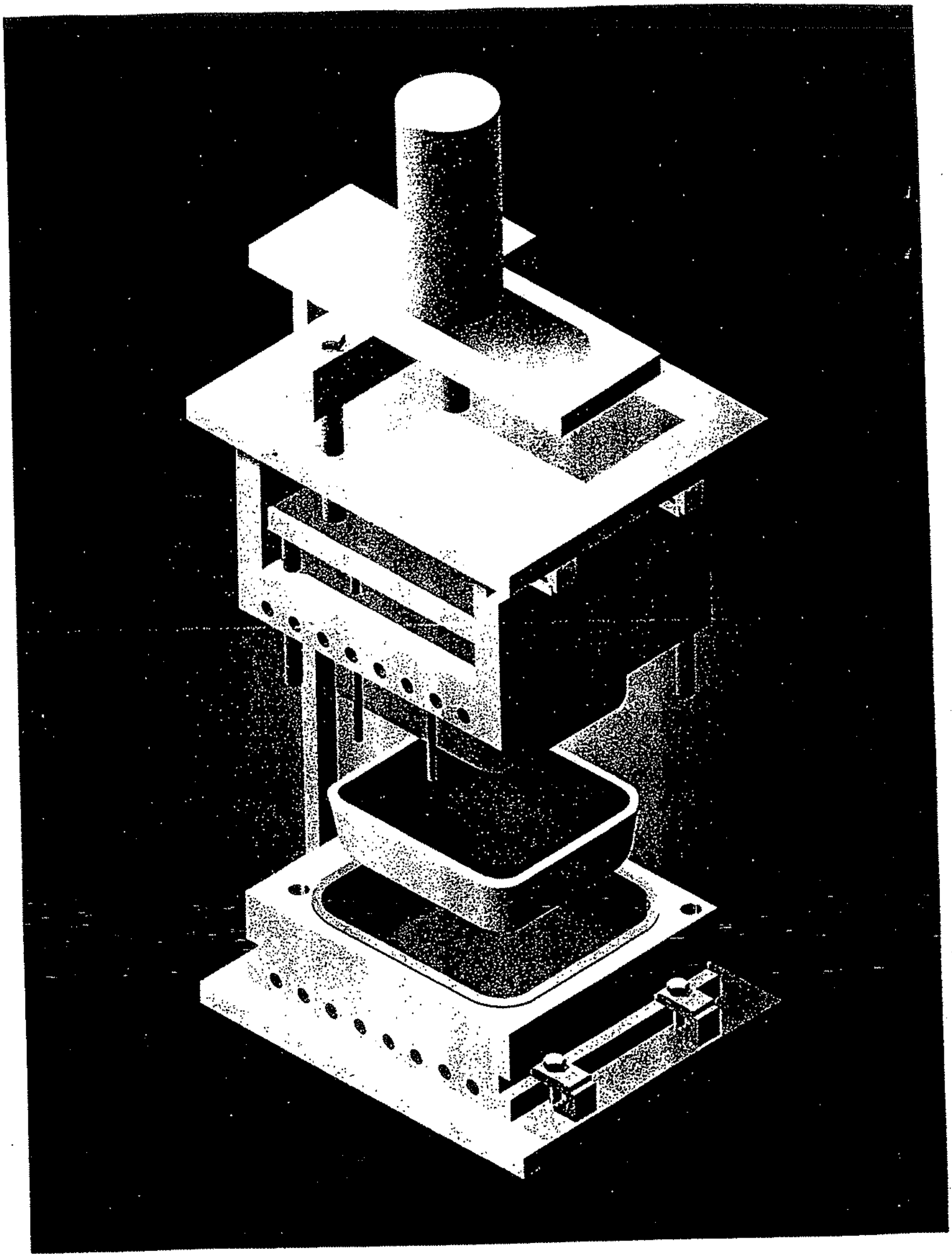
When designing a part, consider design for disassembly (DFD), a concept that is gaining emphasis because of recycling.

Because of recent advances, several methods can be used to recycle polyurethane materials, depending upon the type of material. Most polyurethane resins can be granulated and ground into powder for use as a filler in new parts. The amount of filler that can be used will be based upon your final part requirements. Granulated elastomeric material can also be compression molded under high pressure and temperature to produce new parts. Parts made this way may retain their original elongation and over 50% of their tensile strength.

Glycolysis, a new way to convert polyurethane materials back to their original raw materials, is also showing great promise. Polyurethane materials can be converted into energy: the heat of combustion for RIM polyurethane materials is between 12,000 and 15,000 BTUs per pound, approximately the same as oil or coal. Talk to your Bayer representative for the latest information on polyurethane recycling.



CW 0308724

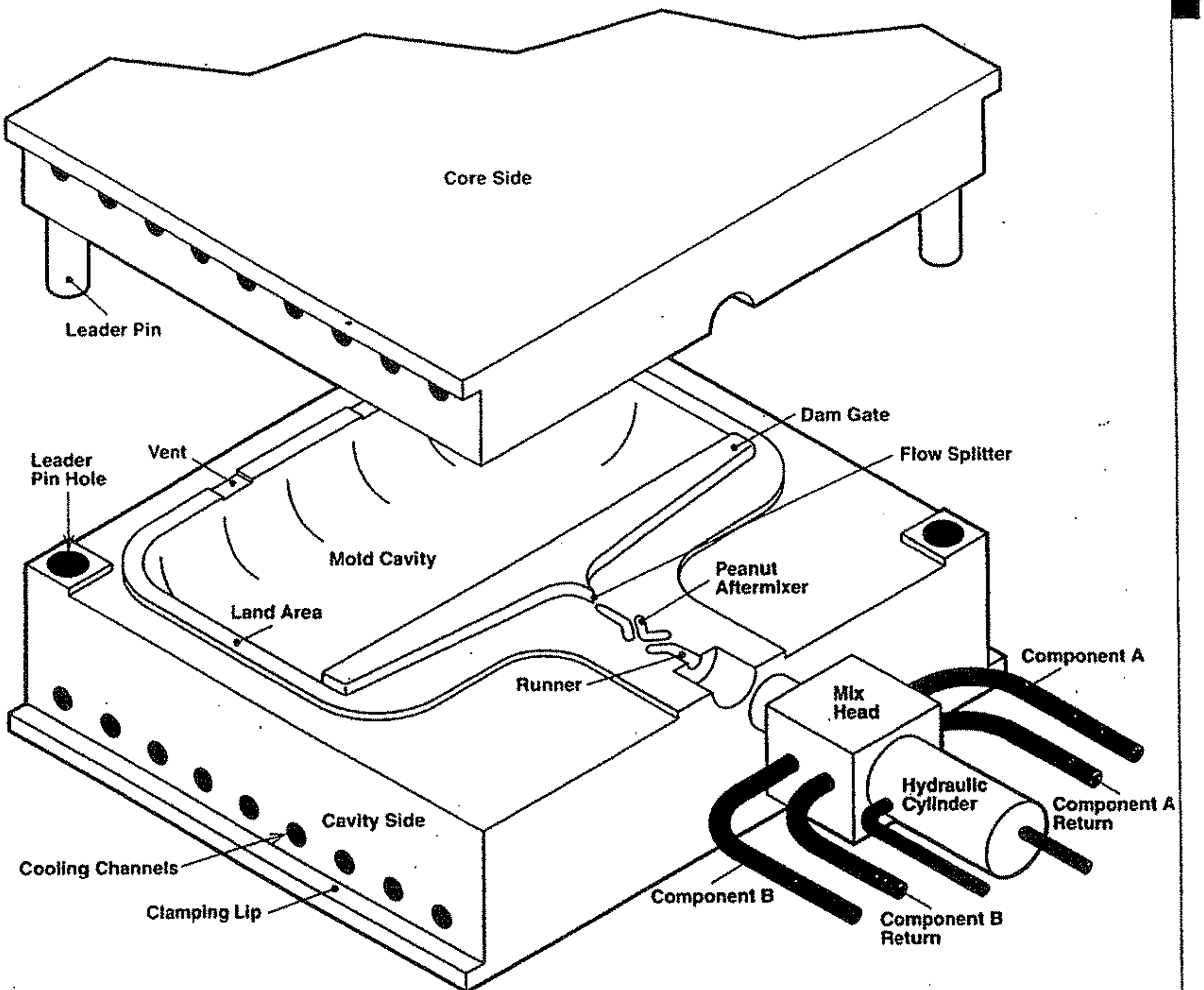


MOLD DESIGN

To make good polyurethane parts, you must have a good mold. A correctly designed mold is the single most-important factor in gaining maximum productivity, uniform part quality, and trouble-free production (see figure M-1). Improvements in gate, mixing head, and aftermixer designs are continuing to add to part quality and uniformity. Molds are sometimes referred to as "tools." In this manual, we use "tools" and "molds" interchangeably.

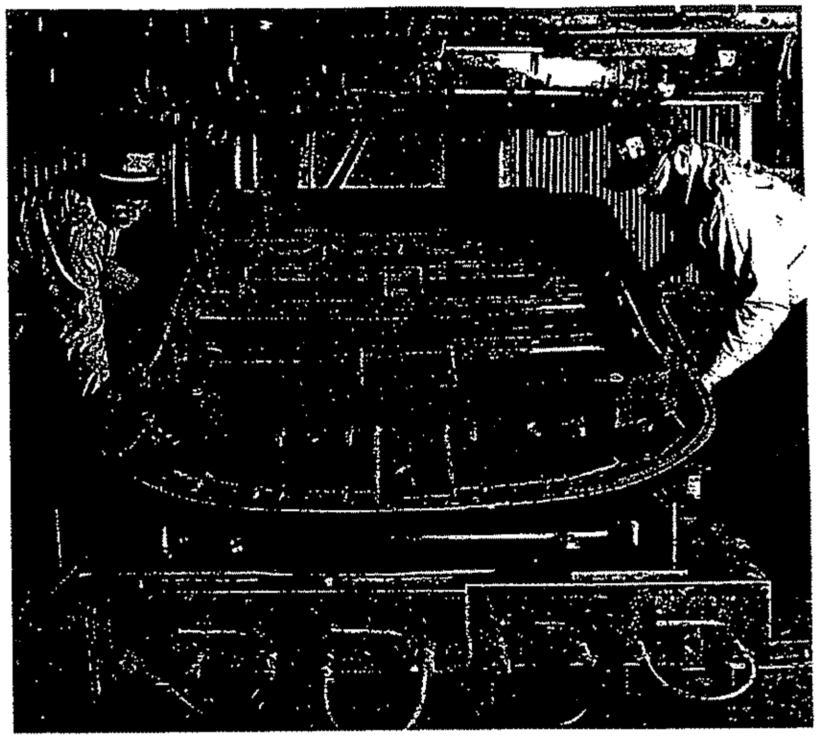
This section of the manual provides guidelines to help you successfully design and build molds, offering some practical rules of thumb. It begins with a discussion of general mold-design parameters, followed by suggestions for gate and parting-line positioning, mold details, finishing, and special tools. Use the information presented herein as general guidelines. Your mold maker is responsible for producing a functional mold. Contact your Bayer representative for information on your specific mold.

Typical RIM Mold **Figure M-1**



A typical RIM mold configuration with a peanut aftermixer and dam gate.

When designing molds for use with RIM polyurethane systems, you must address several issues, including mold size and cost, clamping pressures, part shrinkage, dimensional tolerances, and part repeatability. These general considerations are discussed in this section.



PART SIZE/CLAMPING PRESSURE

With RIM materials, there are no upper size limitations on parts, other than equipment capabilities. Because of metering-machine capabilities, the minimum part weight or shot size is approximately 0.5 pounds (225 grams). For example, this weight roughly correlates to a 6-inch square with a thickness of 3/8 inch at a density of 64 lb/ft^3. While specialty machines to make small parts are available, typical gating and aftermixer requirements would cause too much waste to make smaller parts practical or economical.

RIM polyurethanes are ideal for large parts, with parts heavier than 100 pounds having been made. If one metering machine cannot fill the mold fast enough, in theory two or more can be connected to a mold. Because of practical press-size and other equipment limitations, consider redesigning parts that exceed 50 pounds into smaller components for later assembly. For modular designs, two or more components of an

assembly may be produced from the same mold. Practical limitations on part size include:

- The capacity of the metering machines and mixing heads
- Press capacity and clamping pressures, if self-containment isn't possible
- Gel-time and cream-time limitations

Table 7-1 lists typical molding pressures for RIM systems. Notice molding pressures are more than an order of magnitude smaller than those used in thermoplastic injection molding. Make sure that clamping pressures for your part's projected area exceed the molding pressures.

Table 7-1 Typical Molding Pressures

System	Typical Molding Pressure* (psi)
Baydur® Structural Foam	100
Bayflex® Elastomeric Solids	100
PRISM® Rigid Solids	100
Baydur® STR Composite	200
Baydur® GS	100

* Based upon projected part area in direction of draw. Clamping pressure must be greater than molding pressure.

Table 7-2 Relative Mold-Cost Comparison

Material/ Fabrication Technique	Relative Cost %
Steel, Machined	100
Aluminum, Machined	80
Nickel Shell, Electro- or Vapor-Deposited	70
Aluminum, Cast	60
Kirksite, Cast	60
Zinc, Spray Metal	40
Epoxy, Cast (Prototyping Only)	30

MOLD COSTS

Because of lower in-mold pressures RIM systems use molds that are less expensive than conventional injection molds. Low-pressure RIM systems can be molded in softer mold-construction materials which are easier to machine. Table 7-2 shows a comparison of the costs of different materials and fabrication techniques for a mold to make a simple part.

To further reduce mold costs, simplify part design and avoid undercuts and other elements that add significantly to mold and postmolding costs. The mold's complexity and construction materials determine the total moldmaking cost and consequently a large share of the eventual finished-part cost.

Other factors that influence mold cost, include:

- Number and type of hydraulic slides
- Number and type of different surface finishes
- Part depth and complexity
- Part tolerances

When designing molds, try to weigh the mold cost against the production volume and the cost of postmolding labor needed to finish the part. Postmolding operations — such as trimming, drilling, bonding, sanding, and painting — can add significant cost to a part. Designing a more complex mold may reduce overall cycle time and postmolding labor. While the mold may cost more initially, it could save money over the production life of the part, justifying the higher initial expense. Generally, molds for relatively flat parts with a minimum draw and without undercuts or special surface treatments have the lowest costs.

Finally, let your mold maker know which dimensions are critical and which have looser tolerances. Prioritize them from most critical to nominal. Specified dimensions can influence quotes from mold makers. Using standard fractional inches can be less expensive because a mold maker can use standard machining tools. Specifying decimal-numerical formats with high precision could significantly increase mold cost.

SHRINKAGE CONSIDERATIONS

All plastics, including polyurethane materials, shrink during cooling. Many factors influence the exact amount of shrinkage. Review Bayer Product Information Bulletins (PIBs) for estimated shrinkage percentages for your selected system. Add the shrinkage dimension per inch to every nominal dimension of the part when designing molds. For example, if the shrinkage of a Baydur system is 0.7% and the part is 50 inches long, then the mold's length dimension should be increased 0.35 inches to 50.35.

Also, let your mold maker know which polyurethane system you have selected early in the mold-designing process. Different systems have different shrinkage values, and therefore require different adjustments to mold dimensions to produce your final part. Changing systems or additives during or after mold construction can lead to an increase or decrease in final part dimensions.

DIMENSIONAL TOLERANCES

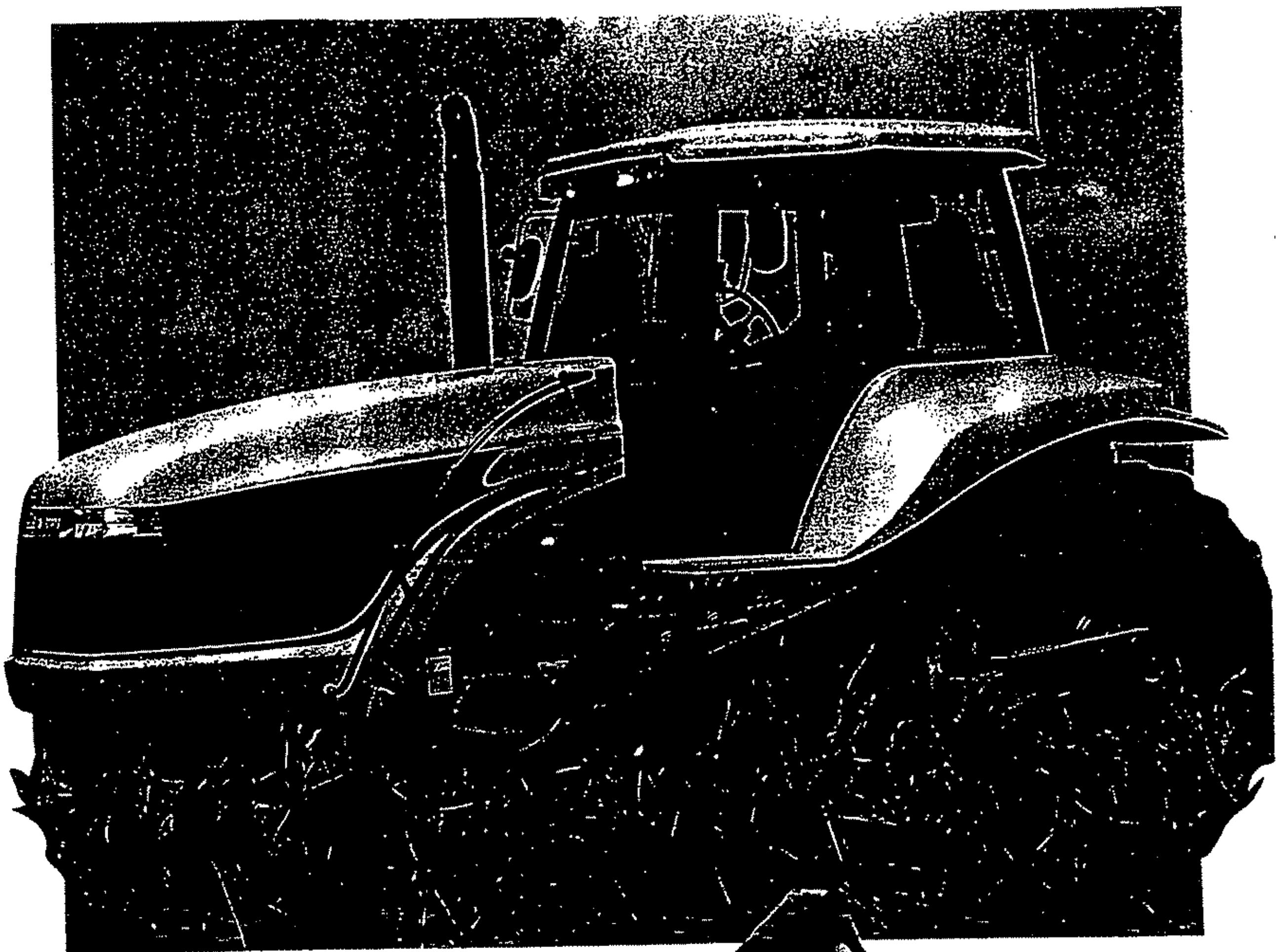
A number of environmental, processing, and material variables determines a part's variation from specified size. Among the most-common variables affecting part-to-part reproducibility are:

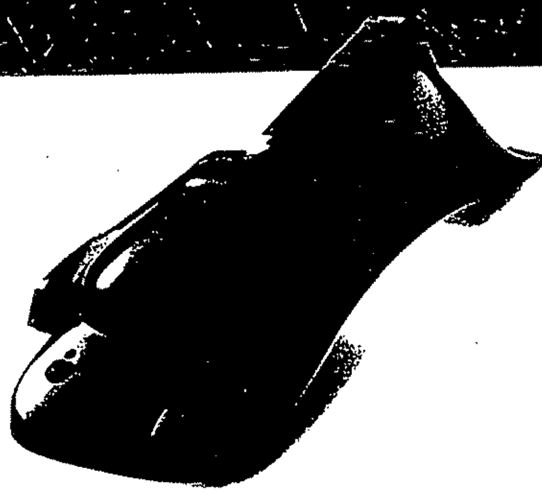
- Part density
- Mold temperature
- Mold alignment
- Mold wear
- Injection rate
- Demolding time
- Component ratios and temperatures
- Ambient temperature and humidity
- Inhibited or uneven part curing
- Mold pressure
- Other production conditions

Because of CAD/CAM and other advances, mold precision has improved and should not be a major concern. Generally, expect a molder to guarantee a dimensional tolerance of 0.1% or less, but part tolerances can be greater.

For best results, discuss all material selection and processing parameters with your mold maker before construction, so that the final tool accommodates your needs.

Often it is more important that your parts fit together properly than that they conform to absolute dimensions. Many times, molders make parts that fit with mating parts, even though both may be slightly out of tolerance. In most cases, this simple matching is functionally satisfactory. If your part must assemble exactly and be accurate to the drawing, explain this to your mold maker as early as possible. To save on costs, adapt your part design to fit within practical dimensional tolerances.

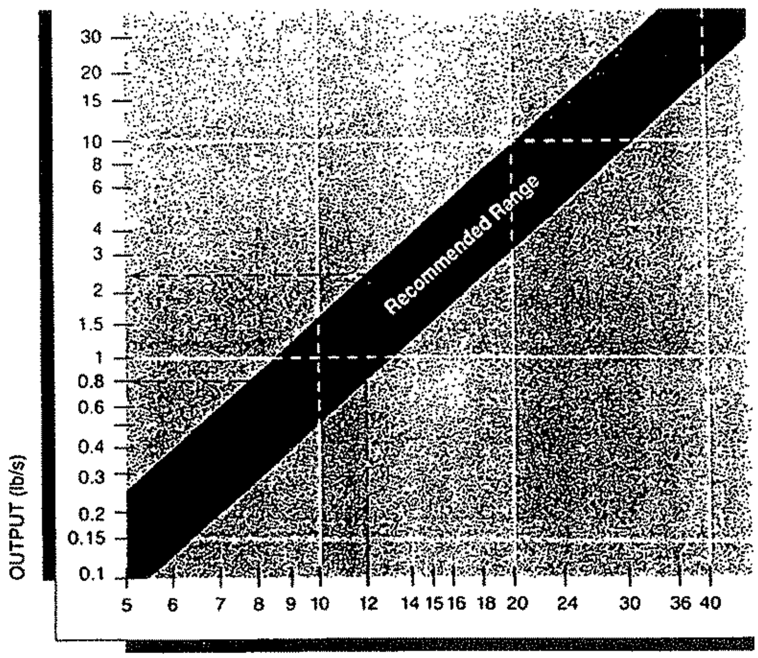




Agricultural tractor fender molded from Bayflex solid elastomer (see cover).

Gating design is a major difference between thermoplastic injection molding and RIM polyurethane molding. Gating is the way the low-viscosity liquid transfers from the mixing head to the mold cavity. Typically there is only one gate per mold in polyurethane molding systems. Throughout this manual we use the word "gating" to refer to the combination of runners, aftermixer, and the gate proper. This section discusses common design and placement parameters. For more information about gating placement, contact your Bayer representative for a copy of our software, RIMgate®. This program helps you determine gate type, placement, and size, as well as good runner and aftermixer design.

Mixing Heads: Diameter Versus Output **Figure 8-1**



MIXING HEAD

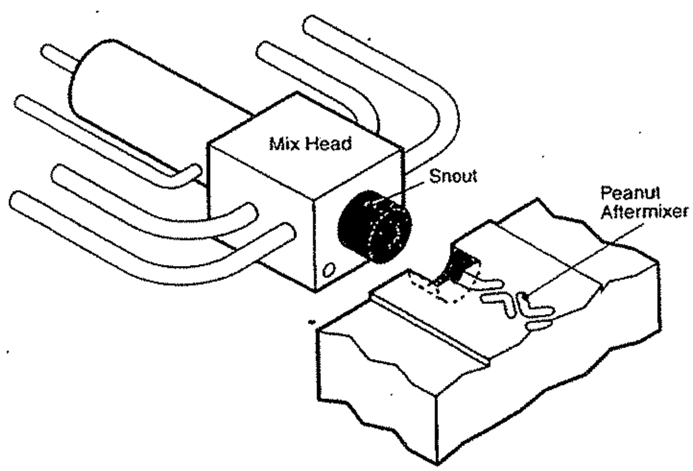
A critical part of molding, the mixing head is the area in which the isocyanate and polyol combine to form a liquid polyurethane material just prior to entering the mold. Self-cleaning mixing heads allow polyurethane materials to be used in large-scale, automated production runs. Mixing heads come in different sizes, each with a range of flow capacities (see figure 8-1). Material flows through opposing injector nozzles in these high-pressure heads, usually at impinging pressures of 1,500 to 3,000 psi (10 to 20 MPa). In the mixing chamber, material reaches ultra-high velocity prior to entering the aftermixer and mold cavity.

Generally, the maximum output of a given mix head equals three times its minimum. For example, a 12-mm mixing head (i.e., the inside diameter of the mixing chamber or outlet tube) can be used for material outputs between 0.8 and 2.4 lb/sec (0.4 to 1.1 kg/sec).

Various equipment suppliers make high-pressure impingement mixing heads. Because the mixing heads vary in size and have different bolt patterns to connect them to the mold, the equipment supplier should have the necessary dimensional information for the specific mixing head you are considering. For more information on mixing heads, contact Hennecke Machinery at 412 777-2000.

Cylindrical Mix-Head Snout, Flush with Mold

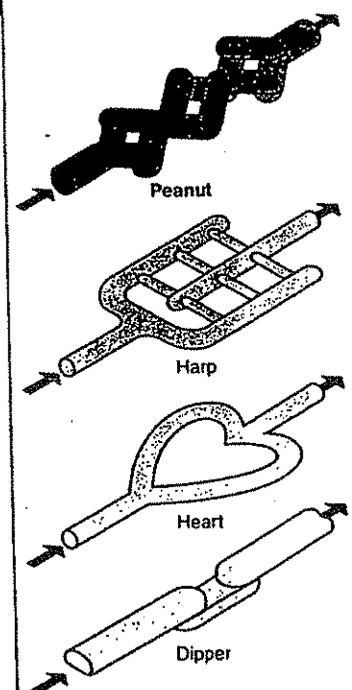
Figure 8-2



molding cycle. the molded peanut shape (sliced in half) visually indicates the quality of mixing, which can help to determine mixing problems.

When cutting a peanut aftermixer in both mold halves, make identical cutting patterns, not mirror images, to prevent blind alleys that can trap air. The aftermixer should be cut into the mold itself or cut into a plate which inserts into the mold. Keep it as close to the mixing head as

Figure 8-3



Types of aftermixers.

Make sure that the mixing head is tightly fastened to the mold during filling. A loose-fitting mixing head may allow the high-velocity stream to bring air into the mold cavity, leading to defective parts.

Most commonly, the head mounts to the side of the mold, parallel to the parting line. Mounting on the side of the mold gives you the greatest access to the mixing head, as well as more flexibility when determining its location. Always try to mount mixing heads on the stationary half of the mold to minimize hose movement. Side-mounted mixing heads are often the only choice for a press with closed platens and limited daylight. Usually, part and mold design and the available press determine the final position of the mixing head.

Most mixing heads have a cylindrical snout, which can be flush with the mold side at the parting line. When possible, design the mold so that the snout intrudes into the mold to create an additional sealing surface when the mold closes (see figure 8-2).

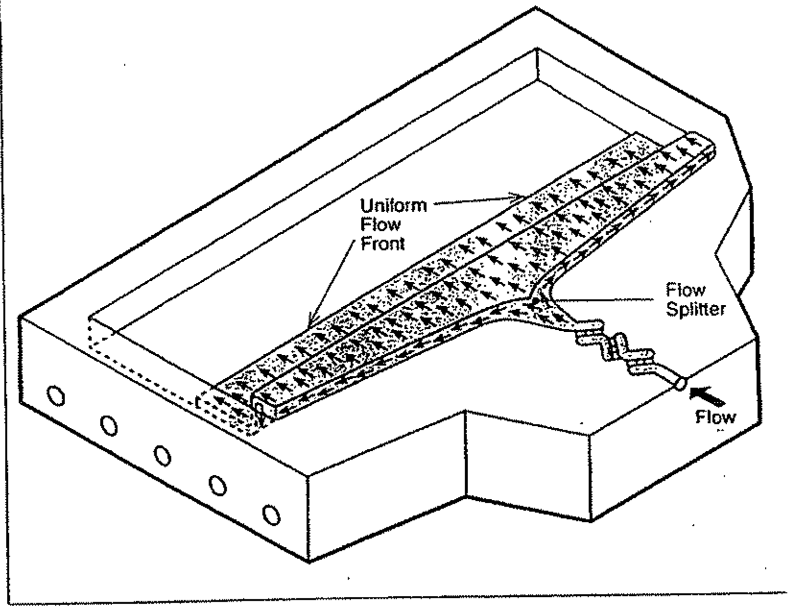
AFTERMIXERS

Any deviation in mix quality can cause imperfections. To ensure complete mixing, use an aftermixer. While many aftermixer designs are available, the "peanut" aftermixer has become the preferred choice (see figure 8-3). Because the mold designer can select the number of V-shapes in this aftermixer design, the peanut aftermixer complements mixhead performance. For instance, increasing the number of V-shapes will help mix difficult-to-mix materials. Additionally, it can be used with any runner diameter. This design causes the mixture to churn as it passes through it, to ensure complete mixing. After the

possible to minimize gate waste. Because of factors concerning temperature control and sealing problems, do not use gate blocks, in which the aftermixer is cut into a separate block and externally attached to the mold. If your mold requires a separate block, place heating channels in the block to control temperature. To adequately seal the aftermixer assembly, the block should be fully supported in the mold.

Over the years, a wide variety of aftermixers have been developed. Although molders tend to have their own preferences, certain types of aftermixers have caused processing problems. The "harp" aftermixer, for example, may have blind areas which trap air. After filling, the trapped air may expand, causing bubbles in the stream, leading to defective parts.

Figure 8-4



Typical dam gating.

EDGE GATING

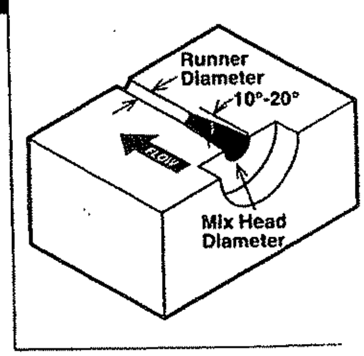
Always edge gate molds for RIM polyurethane systems, unless you are using a Baydur STR composite, which should be center gated to prevent the glass mat from moving. The mixture should enter the mold cavity as a laminar stream, flowing onto the mold wall when entering the mold (see figure 8-4) to avoid air entrapment, the single largest cause of defective parts. It should not be directed into free space or perpendicular to a wall or other obstruction to avoid splashing and resulting air entrapments.

The runner that is adjacent to the mixing-head exit should have the same diameter as the mixing head. When reworking a mold, the runner diameter may not match the new mixing-head diameter. To prevent bubbles and cavitation, make a smooth transition with no sharp edges (see figure 8-5).

The type of material (solid or foamed), mixing-metering machine output, and available room at the parting line determine the gate dimensions. To determine the correct gating configuration, you must know the highest expected machine output. The speed at which material enters the mold must also be kept within limits.

- The upper limits for entrance speeds are 5 ft/sec for foamed systems and 25 ft/sec for solid systems.

Figure 8-5 **Runner Transition**



- Determining lower-limit entrance velocities is difficult, because of the material's reactivity.

- If the injection rate is too low and the gate is too long, highly reactive materials may begin to gel before reaching the mold.

Gates can be designed as part of the mold or attached as a separate block. Again, these blocks can be difficult to seal, especially when the highest pressures are located in the block, and are difficult to heat.

Fitting gate length to the part periphery and balancing flow lengths generally limit the choice of gate locations.

Considerations for gate placement include:

- Locate gates at the lowest possible point in the mold.

- Try to locate the gate on a straight parting line section.

- Try to locate the gate in a position that minimizes the longest flow length.

- Position the gate on non-aesthetic or less-noticeable area.

- Position the gate so that flow will be parallel to designed-in ribs.

- Place the gate closest to cutouts or most-detailed sections. This allows knit lines to form early in the flow.

Table 8-1 Sample Dam Gate Length Calculations for Solid and Foamed Systems

	Solid Systems	Foamed Systems
Known Variables		
Weight Output, O_w	3 lb/sec	3 lb/sec
Density, D	70 lb/ft^3	70 lb/ft^3
Maximum Injection Velocity, v	15 ft/sec	5 ft/sec
Gate Thickness, gt	0.1 in	0.08 in
Calculated Variables		
Volumetric Output, $O_v = O_w / D$	0.043 ft^3/sec	0.043 ft^3/sec
Minimum Cross-Sectional Area, $A = (O_v \cdot 144) / v$	0.41 in^2	1.23 in^2
Minimum Gate Length, $L = A / gt$	4.1 in	15.4 in

Foamed Systems

When using foamed, self-skinning materials, keep the gate thickness as thin as possible so that it can be easily removed from the demolded part. The material in a thin gate will cure to solid material. When it is removed, no foam core should be exposed.

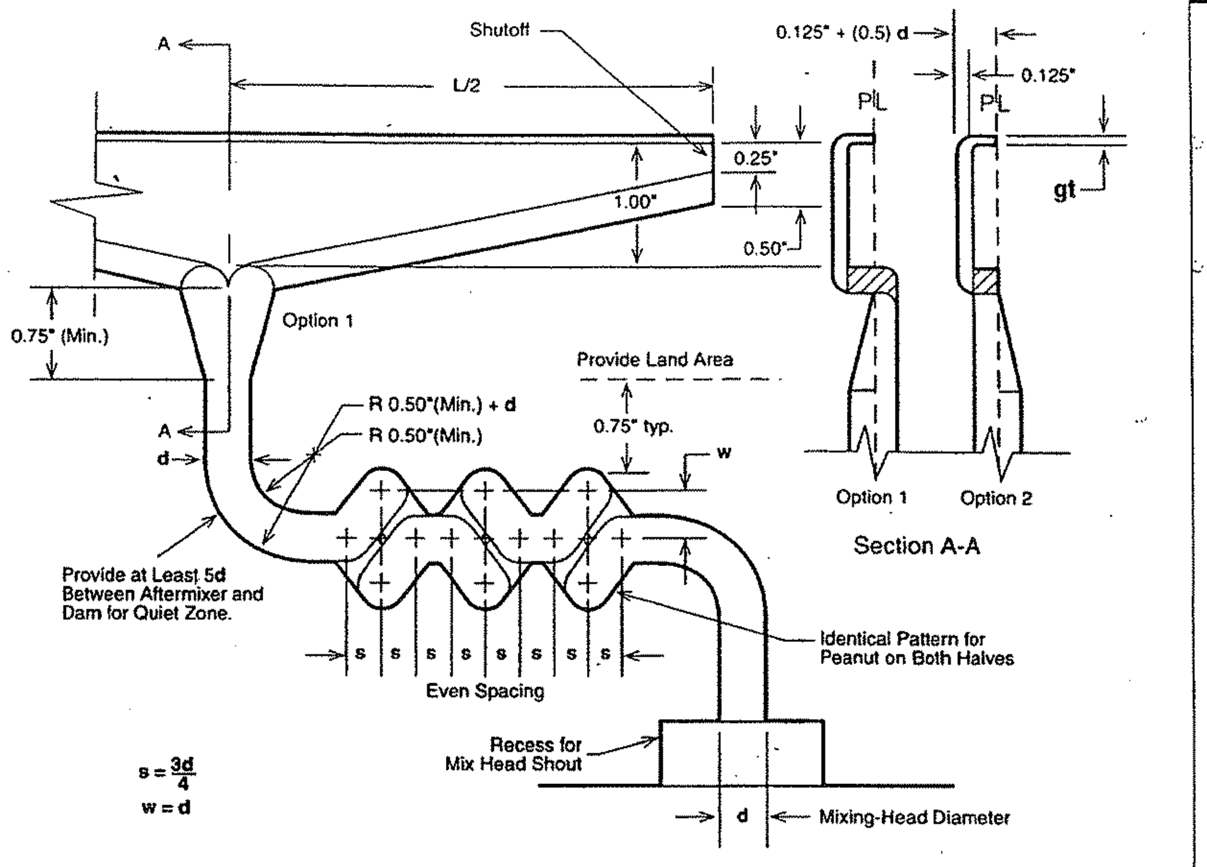
For foamed systems, the rule of thumb for gate dimensions is 6 inches of gate length, with a gate thickness of 0.060 to 0.080 inches, for every pound-per-second output. Maximum stream velocity should not exceed 5 ft/sec. Table 8-1 shows calculations to determine gate lengths to produce a 10-pound part made of different polyurethane systems. Figures 8-6 and 8-7 show complete gate and runner dimensions for dam gates used with foamed systems.

Dam Gates

A dam gate equalizes material flow over its length (see figure 8-7). The gate has a splitting nose that divides the runner into two branches behind the dam. This triangular configuration ensures uniform distribution across the gate. Dam gates are strongly suggested for rigid RIM materials. Table 8-1, and figures 8-6 and 8-7 show a typical calculation for gate length.

Usually gate length and thickness are mutually adjusted to keep entrance velocity from exceeding specified limits.

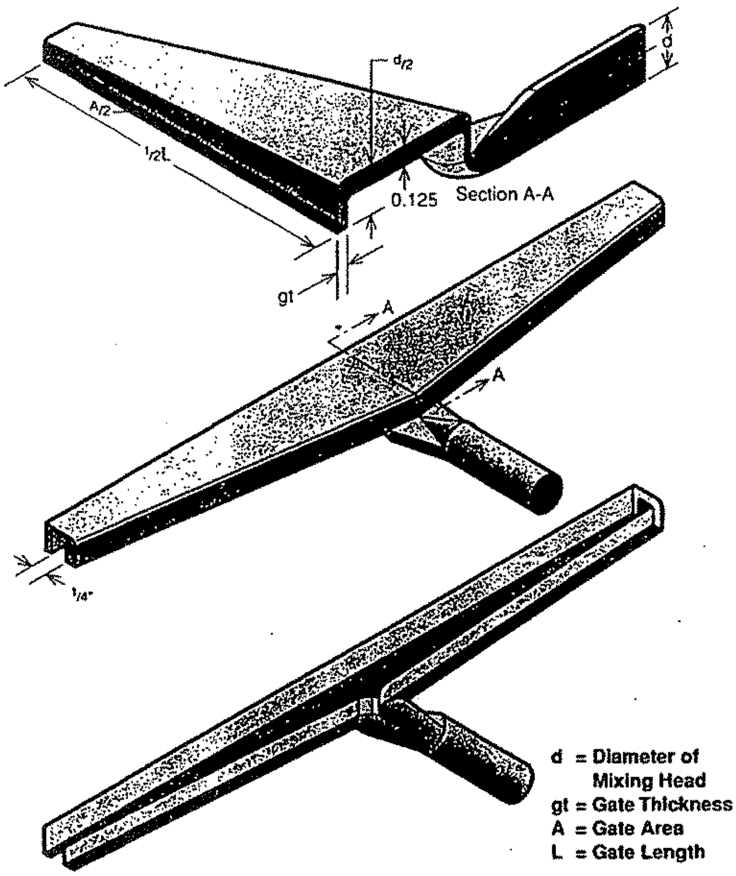
Dam Gate **Figure 8-6**



Solid Systems

When using solid systems, the gate thickness may approach the part thickness. Typically, the maximum stream velocity should not exceed 25 ft/sec. If there are no sharp curves in the flow path between the gate and main cavity and if the part and gate thickness are equal, stream velocities exceeding 100 ft/sec have yielded acceptable automotive parts using fan gates. PRISM solid rigid polyurethane systems can be edge gated or direct filled.

Figure 8-7



Dam gate dimensions (option 1).

Fan Gates

The two most-common types of fan gates (see figure 8-8) are the straight-sided or triangular gate and the preferred quadratic gate, which has a parabolic profile. In fan gates, the runner gradually flattens and bends in the wall direction. The triangular gate's apex angle should not exceed 40° to avoid cavitation in the stream that could generate bubbles. With this angle limitation, thin gates can become very long, particularly for large parts. Long gating can lead to excessive waste.

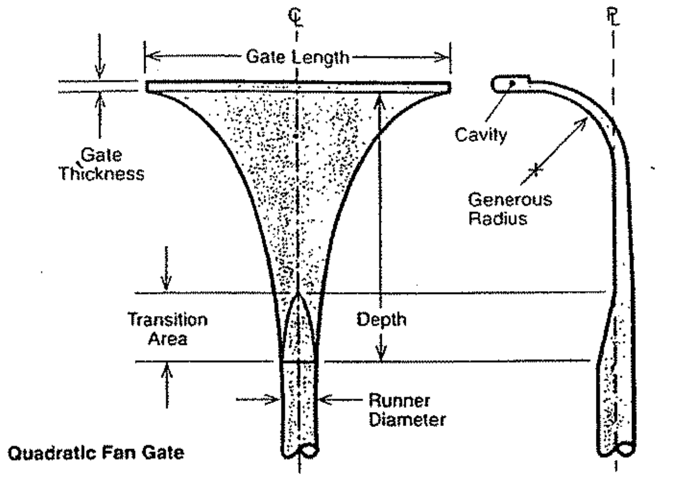
- As a rule of thumb: practical maximum gate length is 6 inches.

- Fan gates are recommended for solid elastomeric systems that have a fast curing time.

In practice, fan gate length averages four inches. Usually gate length and thickness are mutually adjusted to keep entrance velocity from exceeding specified limits. Table 8-2 and figure 8-9 show relationships for determining fan gate dimensions.

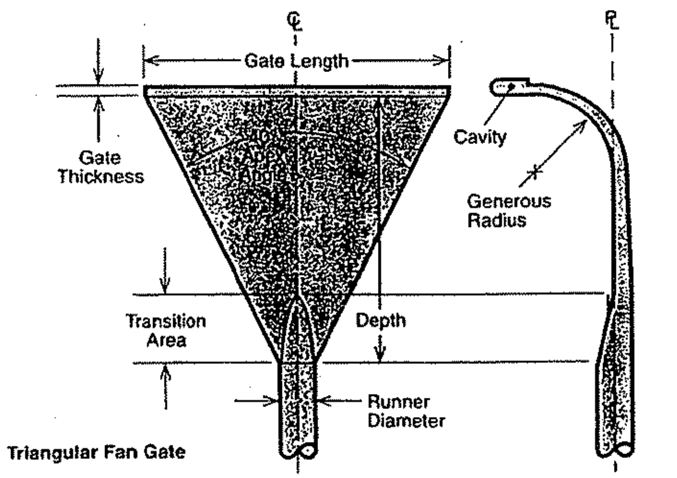
CW 0308737

Fan Gates **Figure 8-8**



Quadratic Fan Gate

Gate thickness decreases linearly from the transition area to the entrance of the cavity.



Triangular Fan Gate

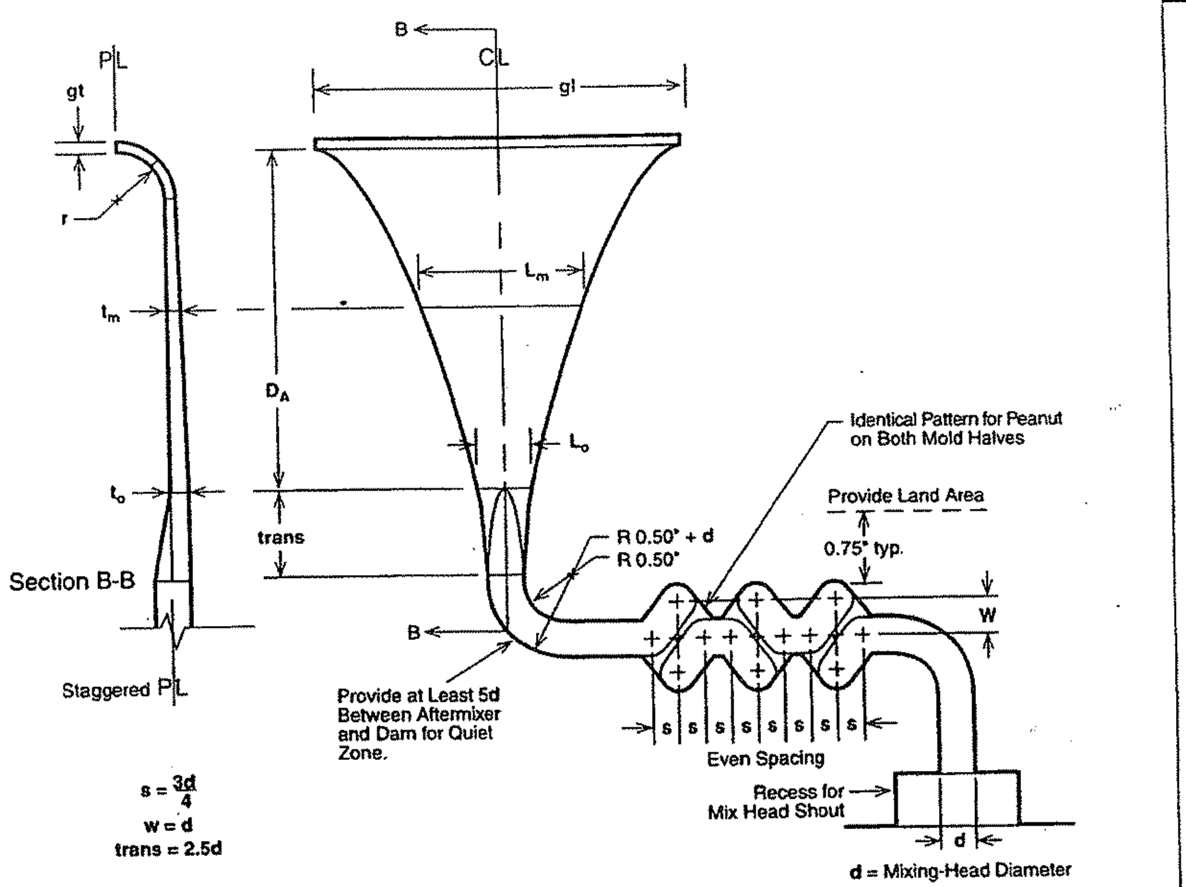
Gate thickness decreases quadratically from the transition area to the entrance of the cavity.

Table 8-2 Calculation of Typical Fan Gate Dimensions

O_w	=	Weight Output, lb/sec
O_w	=	Part Weight/Shot Time
D	=	Liquid Density, lb/ft^3
O_v	=	Volume Output, ft^3/sec
O_v	=	O_w / D
v	=	Velocity, 5 ft/sec Foam 25 ft/sec Solid
A_f	=	Final Fan Area = $[O_v / v] \cdot 144$, in^2
gl	=	Gate Length (Specify) Typically gate length depends on the available space along the parting line at the gate location.
gt	=	A_f / gl, in
A_o	=	Initial Fan Area, in^2
A_o	=	$\pi d^2 / 4$
t_o	=	d/2
L_o	=	A_o / t_o
A_m	=	Mid Fan Area, in^2
A_m	=	$(A_f + A_o) / 2$
t_m	=	$(t_o + t_f) / 2$
L_m	=	A_m / t_m
D_1	=	gl•(2π/4) [illegible]

CW 0308738

Fan Gate Dimensions **Figure 8-9**



BALL CHECK

In cases where the mixing head is not attached to the mold, use a ball check to prevent the mixture from running out of the mold before gelling (see figure 8-10). Typically, ball checks are suggested when using hand-held mixing heads, also known as "handguns," for filling multiple molds mounted on a carousel, or when molds are positioned in a half-circle.

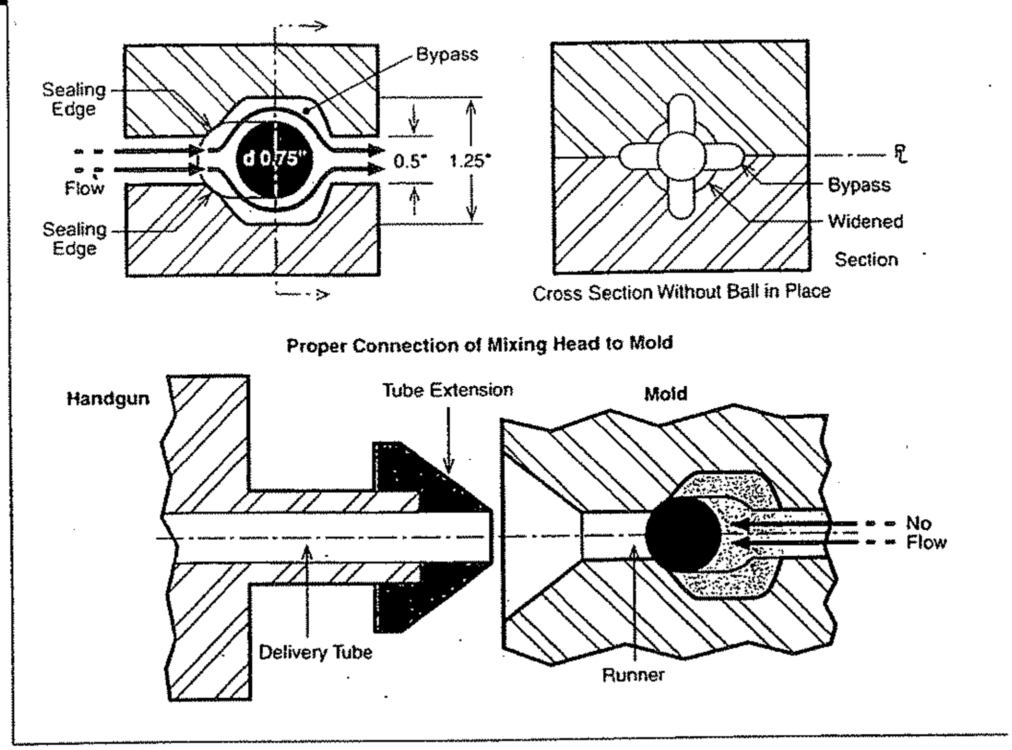
The ball-check design allows flow in one direction only. In this design, a rubber ball can freely move in a channel, with one end of this channel having bypasses to allow the liquid to flow around the ball. When filling ends, internal pressure pushes the ball to the other end of the widened section where there are no bypasses. The ball then seals the channel, effectively preventing back flow.

CENTER-GATED DIRECT FILL

When designing molds for use with structural RIM materials, always plan on direct fill. Centering the gate prevents the liquid from pushing the glass mat out of position and allows for uniform flow in all directions. In this filling method, the mixing head attaches directly to the mold wall, creating an airtight seal.

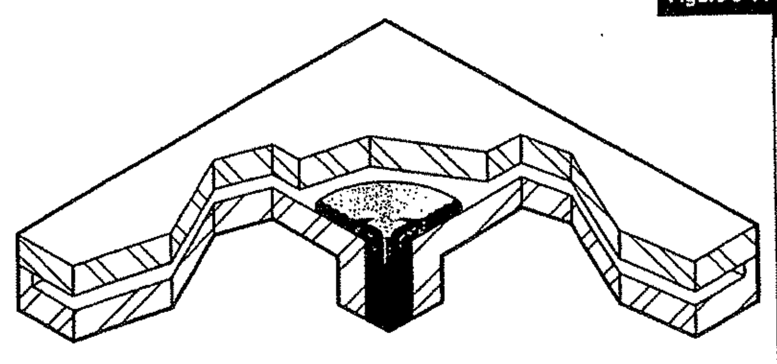
CW 0308739

Figure 8-10



Ball check.

Figure 8-11



Direct fill into center of mold.

A center gate is particularly important in parts made of structural RIM systems, such as Baydur STR composite, where back pressure is more likely. For this type of system, center-gated direct fill offers the simplest, cheapest, and most straightforward type of gating. It minimizes flow lengths and gives more uniform flow inside the cavity (see figure 8-11). Additionally, with a self-cleaning mixing head mounted flush in the cavity wall, gating waste is minimized, if not eliminated. Because the mix head is located in the center of the mold, leaking is also minimized.

Direct fill also has disadvantages: it may cause a blemish opposite the entry location in the part because the mixture makes a 90 degree turn over a sharp edge (see figure 8-12). This extremely unfavorable flow could cause bubbles and scarring.

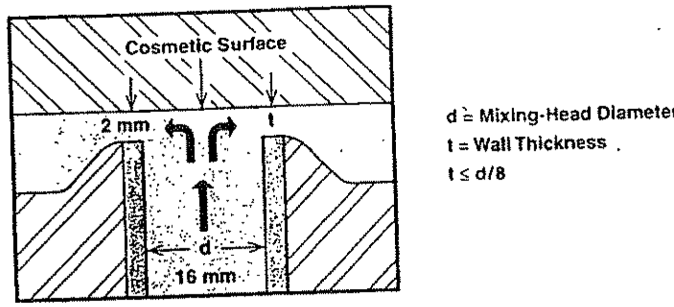
- Research shows that the mixture will be bubble-free only if the wall thickness at the entry point is less than one-eighth of the entrance-area diameter (see figure 8-13).

For example, a 16-mm entrance diameter should not have a wall thickness larger than 2 mm (0.08 in). Because most parts are thicker than this, redesign the wall thickness near the entrance, narrowing it to this value (see figure 8-13).

If the mixing head cannot be flush-mounted, use a short sprue. Keep it as short as possible, so that mold release can be sprayed into the sprue cavity.

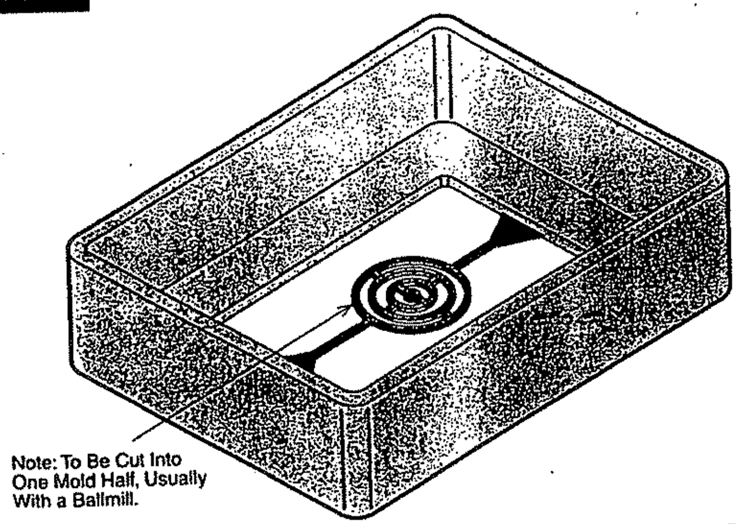
For noncomposite systems, the center-gate position makes using aftermixers difficult. When the part has cutouts large enough for an aftermixer and edge gate — such as a picture frame — an aftermixer can be used. Use a peanut aftermixer or one made of concentric channels with a staggered "spoke" design to provide a labyrinth. This configuration splits and reimpinges incoming material before it enters the mold cavity (see figure 8-14).

Figure 8-13



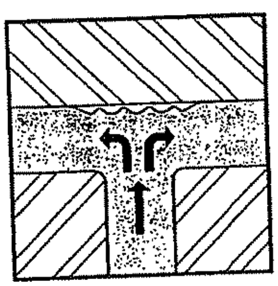
Ratio of wall thickness to mixing-head diameter for direct fill.

Figure 8-14



Typical circular aftermixer.

Figure 8-12



Gate marks caused by poor gate dimensioning.

CW 0308741

Many times, part configuration limits parting-line location. However, the position of the parting line directly influences many other mold features, including gate position, mold tilting, and venting (see figure 9-1). Preferably, molds should fill from the lowest point to the highest, with the parting line as high as possible to accommodate vents and prevent air entrapment (see figure 9-2). Try to design parts so that a simple two-part mold can be used. If the parting line cannot be located in the highest position in the mold cavity, use a drag plate to create a secondary parting line (see figure 9-3).

Mold makers or Bayer personnel can help determine venting areas from part drawings or models.

Figure 9-1 Mold in Tiltable Press

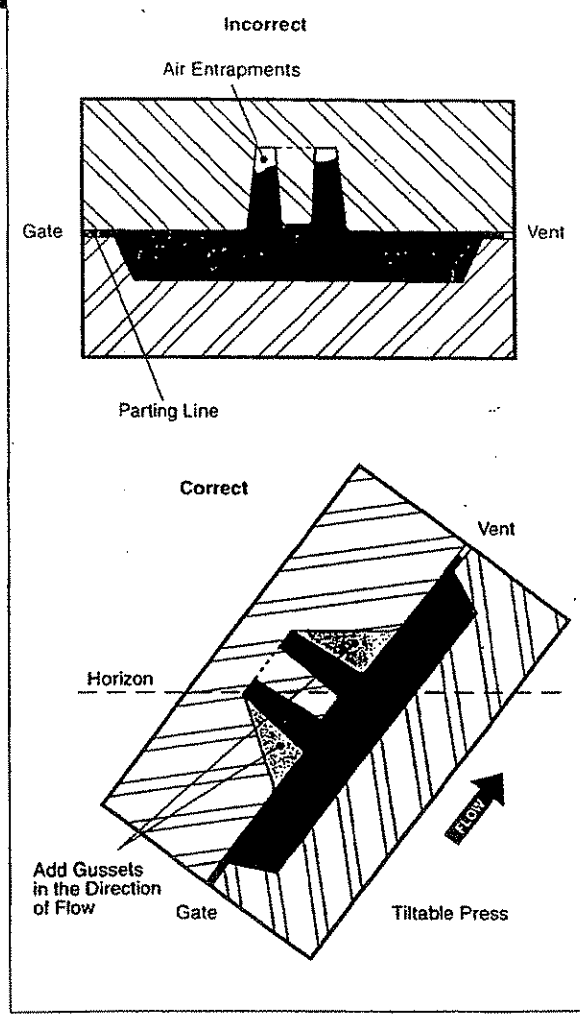
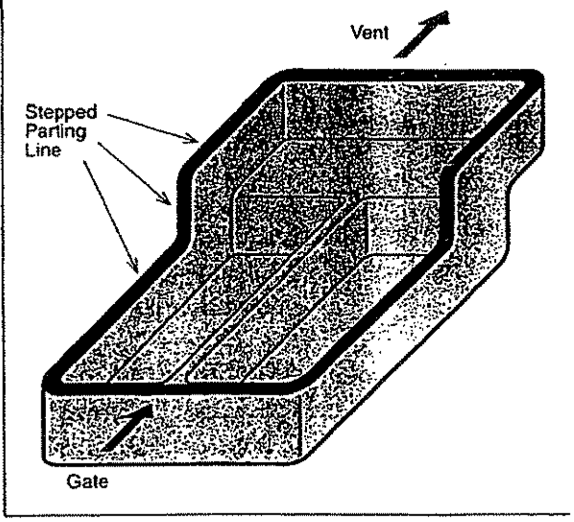
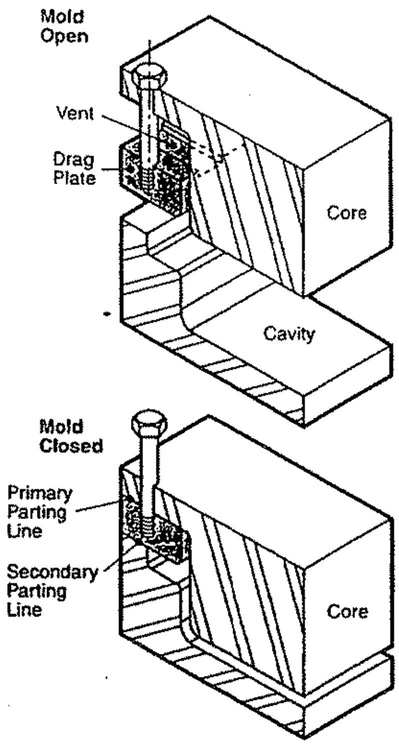


Figure 9-2 Gate Low, Vent High



Secondary Parting Line Via Drag Plate

Figure 9-3



MOLD SEALING

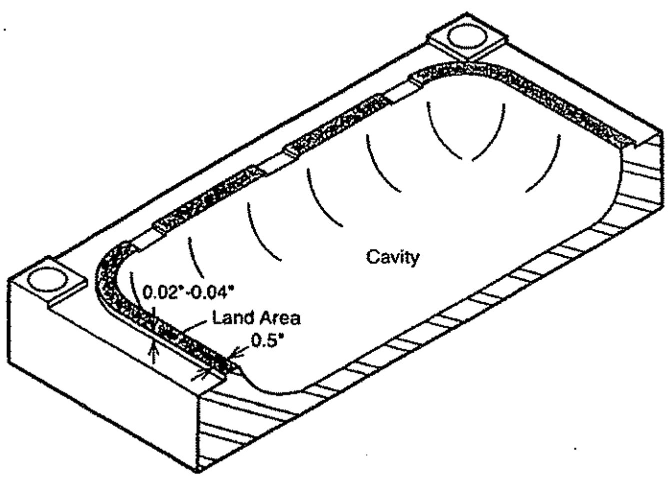
When molding parts made of RIM polyurethane materials, the molds must be adequately sealed to ensure part density and minimize flash (see figure 9-4), the excess material that occasionally forms along the parting line. Sometimes flash is intentionally created in select areas to help fill the mold cavity. In these cases, the mold will have a dump well to collect excess material (see figure 9-5).

Table 9-1 Approximate Sealing Edge Widths

Mold Material	Approximate Sealing Edge Width (inches)
Steel	1/2
Aluminum	3/4
Kirksite	3/4
Nickel Shell	1/2
Epoxy	1

Mold Sealing — Land Area

Figure 9-4



- Make sure that flash does not get into the knockout mechanism. Flash can bind the knockout plate, leading to torn, deformed parts.

Mold seals must be "liquid tight." To achieve this seal, internal mold pressures must not exceed clamping pressures. The sealing edge around the cavity and the runner should be as small as possible to reduce contact area and provide a good seal (see table 9-1).

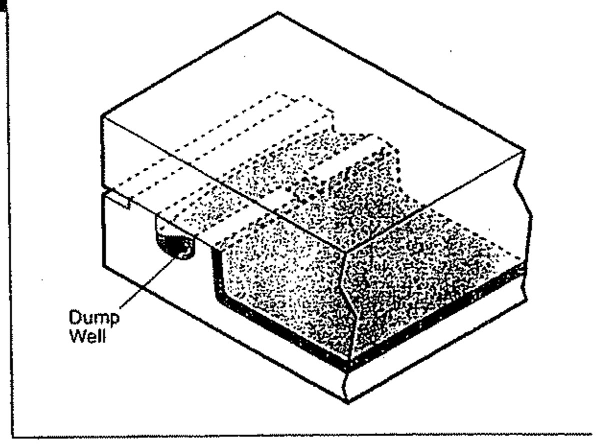
MOLD VENTING

All molds used with RIM polyurethane systems must have vents at the highest point in the mold cavity so that air in the cavity can escape during the filling and molding processes. Maximize the number of vents to aid filling. When using a new mold, cut only a few vents. It is easier to add more vents if necessary than it is to remove unnecessary ones.

- Make vents wide and shallow; not narrow and deep.
- As a rule of thumb, design mold vents that are 1 inch wide, 0.010 inch deep, and 3 to 4 inches apart, edge-to-edge (see figure 9-6).

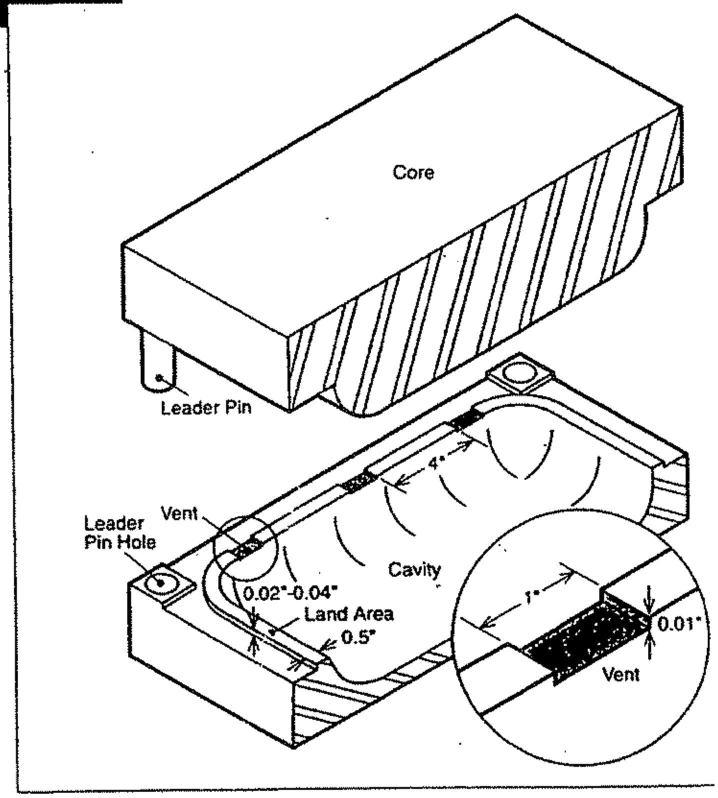
Design ribs and bosses to allow for air displacement. Consider connecting them to a part wall or placing a tiny, tapered hole in or through the mold wall to help venting (see figure 9-7). If a hole is used, it must be accessible from the mold exterior so that it can be cleaned as needed. If necessary, use vent pins that are part of a knockout mechanism to vent bosses. All protrusions above the parting line must be vented separately.

Figure 9-5



Mold dump well.

Figure 9-6 **Typical Mold Venting**



Typical mold venting using 4-inch spacing. Vents on high side of mold.

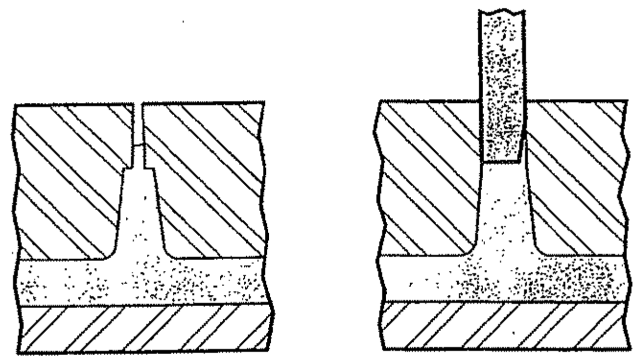
MOLD FILLING

Filling levels vary with systems and specified density. Solid systems fill the mold completely. In contrast, foamed systems usually fill 40 to 80% of the mold cavity, depending upon final part density required.

When planning for proper material flow, consider any obstructions in the mold cavity, such as cores. To reduce the effect of weld or knit lines, liquid polyurethane systems must flow around such obstructions and rejoin. If flow fronts join early in the filling process, weld lines will be unnoticeable and show very little, if any, loss in properties.

Weld-line formation is particularly important in short-fiber, reinforced materials, because fibers tend to align with flow direction. Where the flow fronts join, fiber crossover and homogeneity will not occur. With faster reaction and gelling times, this problem intensifies. To minimize this condition, consider a different gating position closest to the largest obstruction. Computerized mold-flow analysis can help determine where problems may occur. Contact Bayer for more information on filling analysis.

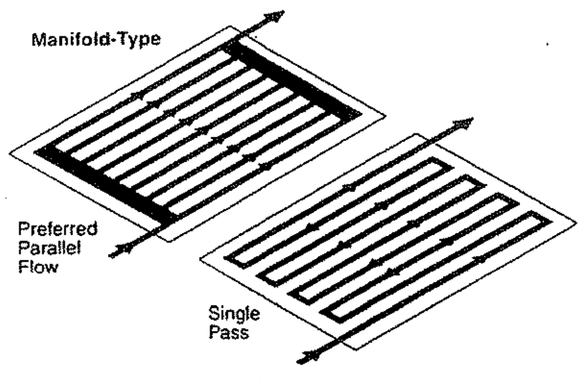
Figure 9-7



Vent solutions for field problems using a tapered hole or knockout pin.

When designing molds, you must account for mold temperature control and cooling line placement. Additionally, this section addresses demolding methods and any special inserts or movable cores in the mold.

Figure 10-1



Mold cooling channels.

MOLD TEMPERATURE CONTROL

Mold cooling directly affects the quality of your finished part. When the "A" and "B" components in a RIM polyurethane system react, they generate heat, as much as 150 BTUs per pound of metered material, depending upon the system used. Mold temperature must be kept at a constant, specified level, usually between 120 and 180°F, again depending upon the system used. To maintain a controlled temperature in the mold, this heat must be conducted through the mold walls, away from the curing part. Cooling channels with circulating water is the typical method for maintaining and controlling mold temperature.

Generally manifold-type cooling lines are preferred. They offer more-even cooling, minimize hot spots, are more efficient, and use more water at a lower pumping-head pressure than single-pass systems (see figure 10-1).

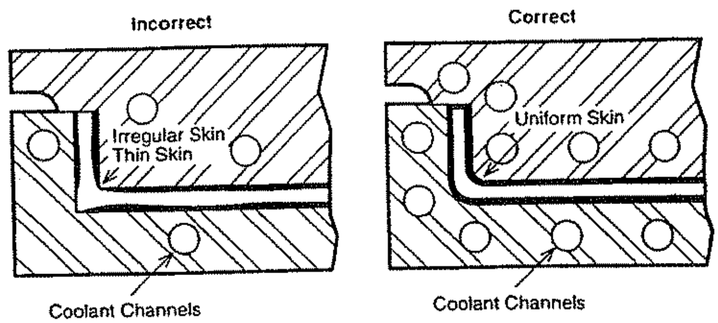
The selected polyurethane system, mold-making material, mold size, and mold complexity determine the placement and number of cooling channels. For best results, cooling channels should be located 1-1/2- to 2-channel diameters from the mold surface and a maximum of 2 inches apart (see figure 10-2). Channels typically have a diameter of 3/8 inch. Pipe or tube fittings on cooling lines should have diameters equal to the diameters of the lines to prevent blockages. Remember to incorporate cooling lines adjacent to inside corners, gate blocks, and other slow-to-cool areas to produce good parts (see figure 10-3). Proper cooling is especially important when using Baydur structural foams which need adequate cooling to form rugged skins.

DEMOLDING METHODS

To help remove parts, use demolding techniques in strategic locations. The three most-common demolding methods are:

- *Mechanical or Hydraulic Knockouts* — have pins in strategic locations to push out parts (see figure 10-4);
- *Air Assists* — have proven sufficient with simple, flat parts (see figure 10-5);
- *Vacuum Cups* — are applied to parts for manual removal, but are rarely used.

Figure 10-3 Effect of Temperature Control on Skin Formation



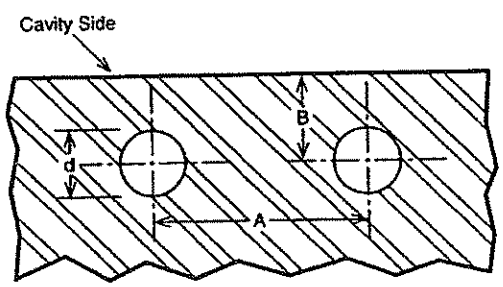
The most-common demolding device, **mechanical knockout pins**, are rods that are flush with the wall during molding and protrude when the mold opens.

Knockouts should not be actuated until the part has achieved sufficient "green strength" or when the part has solidified enough to maintain its shape and be removed from the mold without knockout marks or tears. The part's limited compressive strength at the demolding time requires a large contact area for these devices. Whenever possible, have pins actuate against a rib, corner, or boss to distribute their forces over a wider area of the part.

Small-diameter pins can damage parts made of foamed systems during demolding. Use large pins, with a minimum diameter of 3/8 inch. For flat areas, 1/2-inch pins are recommended. In contrast, solid rigid parts, such as those made of PRISM polyurethane, can withstand actuation by 1/8-inch pins, particularly in areas where ribs or walls intersect.

Pins in a knockout plate are usually positioned in the moving half of the mold. Mechanical stops in the press

Typical Cooling Channel Placement **Figure 10-2**



	Aluminum Mold	Steel Mold
Diameter of Channel (d)	3/8 – 1/2 inch	3/8 – 1/2 inch
Maximum Depth (B)	3/4 inch	3/8 – 3/4 inch
Maximum Distance (A)	2 inches	1-1/2 inches

actuate the plate when the mold opens (see figure 10-4). The press usually has four stops: large screws that must be adjusted so that all four touch the knockout plate at the same time to ensure uniform knockout action to help prevent part binding.

If knockouts are located in the stationary mold half, hydraulic cylinders move the knockout plate. Alternatively, chain-activated knockout plates can be used. Return pins push the knockout pins back to their flush position when the mold closes.

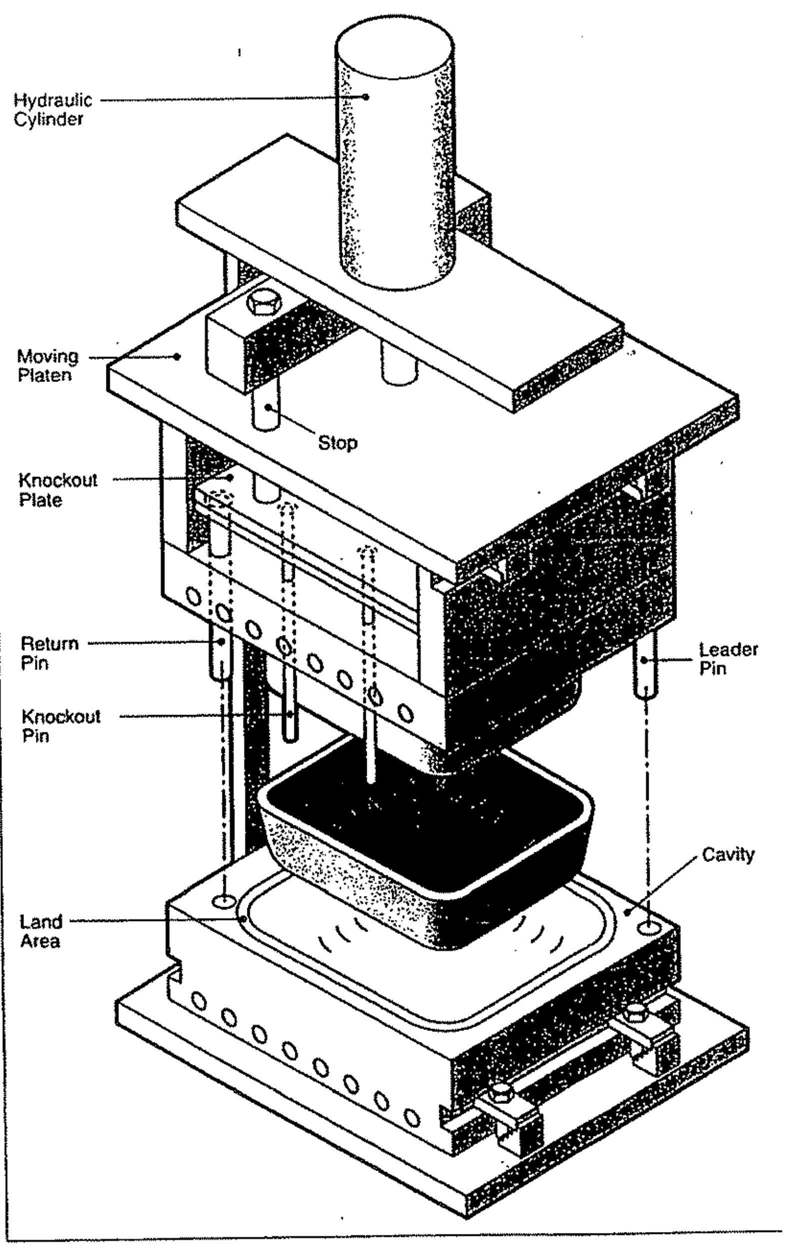
Unconnected knockout pins can occasionally be used if they are actuated with double-acting hydraulic cylinders. Do not use cylinders with spring returns as they may not retract completely when the mold closes.

A combination of mechanical knockout pins and air assists is another common demolding technique. A vacuum may develop between the core and a part as the knockout pins activate. Compressed air blown into this vacuum releases the part (see figure 10-5). The air valve must seal perfectly and be flush with the mold so that air cannot escape and create bubbles during mold filling.

Vacuum cups can be used to remove parts from core sections. Rules of thumb for using vacuum cups include:

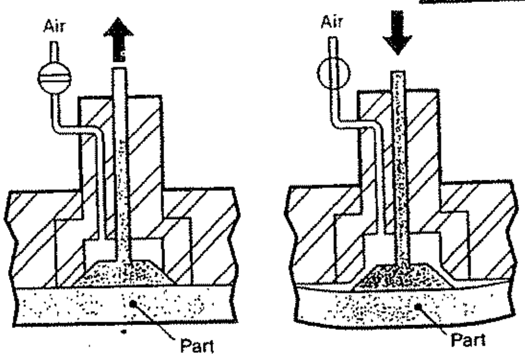
- Distribute large vacuum cups uniformly over the part surface.

Figure 10-4

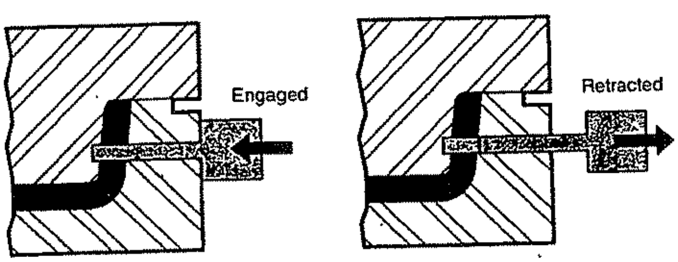


Typical mold arrangement.

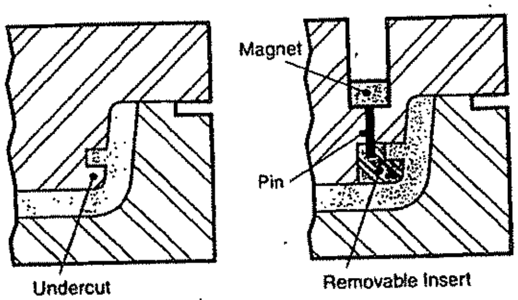
Air-Assist Demolders **Figure 10-5**



Slides **Figure 10-6**



Undercut Molded with a Removable Insert **Figure 10-7**



- Use a quick motion to remove parts, rather than a long pulling motion with slowly increasing force.

MOVABLE CORES AND INSERTS

Use movable cores for parts with undercuts — snap fits, holes, or cutouts — located perpendicular to the direction of draw. Movable cores must be liquid-tight to prevent material flashing into the actuators and locking the slides (see figure 10-6). Consider using O-rings on the pins and actuator shafts to prevent leakage.

Removable inserts are another method for making undercuts (see figure 10-7). Generally molders do not like to use these inserts, as they are labor-intensive, can fall out of position, and may damage the mold. Pins aligned in the direction of draw hold inserts in position, allowing the insert to be removed with the part. Discuss any movable cores and inserts with your mold maker during the mold design process.

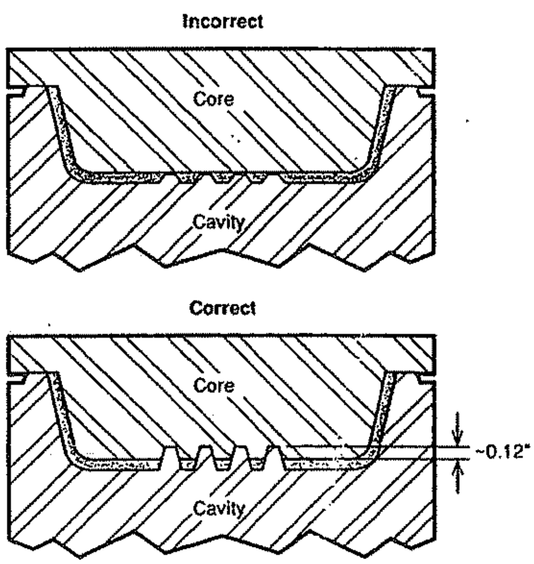
MOLD DESIGN FOR SLOTS

When designing for a part with slots, do not design a mold such that the cores just touch the opposite wall; rather, seat them approximately 1/8 inch into the wall (see figure 10-8). This design practice will facilitate flash removal, because the flash will be perpendicular to the part's surface. If possible, design a mold with a stepped parting line for slots (see figure 10-9).

SHEAR EDGES

Fiberglass mat, used in parts made of Baydur STR composite systems, should fill the entire mold cavity, leaving no empty areas between the mat and mold wall. To meet this requirement, the mat needs to be slightly oversized. The mold will need a renewable-steel shear edge to cut away any glass-fiber overhang, a pocket to hold this excess, and a mold seal external to the shear edge (see figure 10-10). Because glass fibers can erode softer metal molds, we strongly suggest using steel molds for structural RIM.

Figure 10-8



Recommended orientation for slots.

Mold for Louvers Without Side Pulls

Figure 10-9

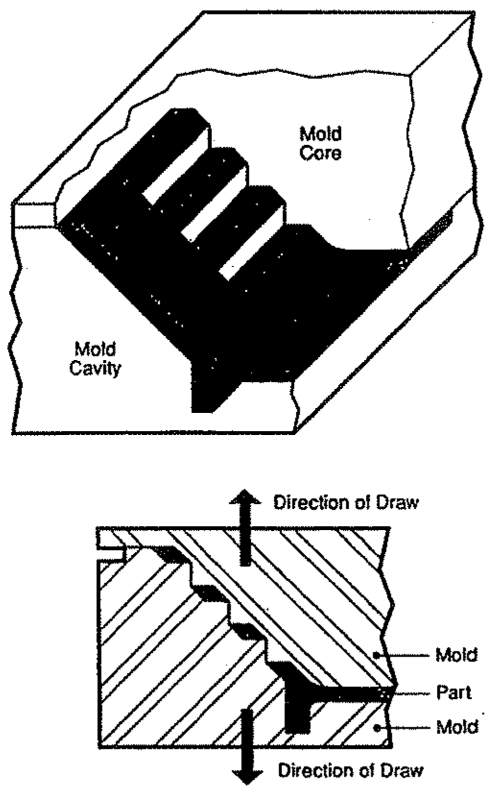
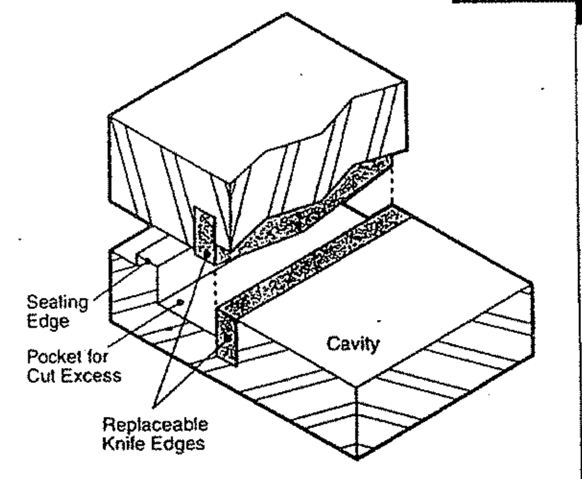
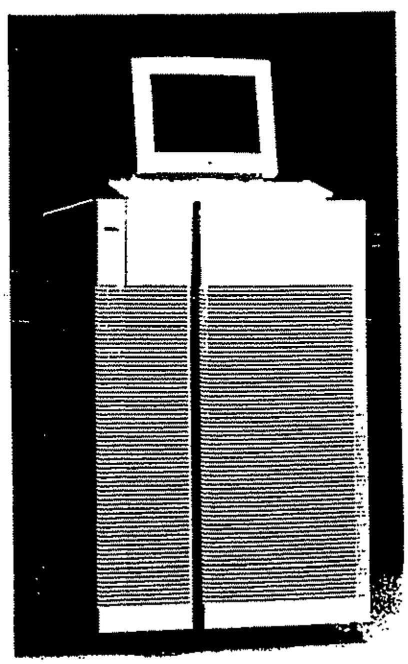


Figure 10-10



Mold shear edge.



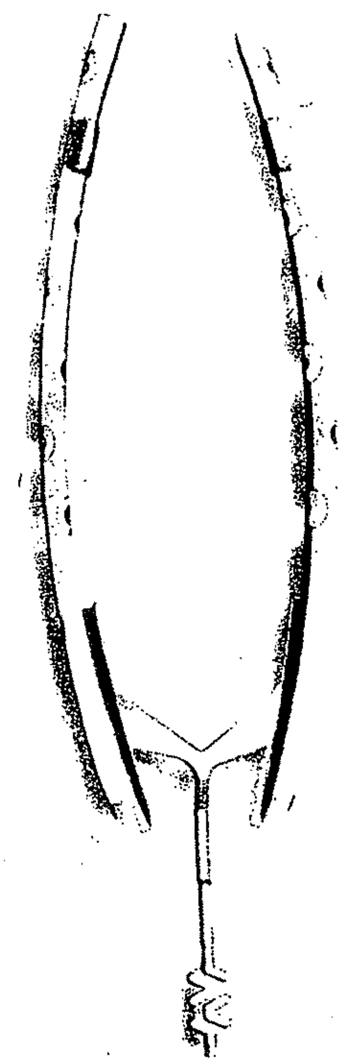
Occasionally, a multiple-cavity or self-contained mold is used for economic or processing considerations. These special molds have unique requirements, which are discussed in this section.

MULTIPLE-CAVITY MOLDS

Molds can be designed with multiple cavities to produce several parts simultaneously (see photo). Multiple-cavity molds rarely have more than four cavities, with a two-cavity mold being more common. A special type of multiple-cavity mold — called a "family mold" — produces mating parts of an assembly. Typically, multiple-cavity molds are economical for larger production runs. While the tooling may be more expensive than single-cavity molds, production time and costs can be lower with multiple-cavity molds.

In a typical multiple-cavity mold, the reaction mixture enters the mold through a centrally located edge-gating system. The fill time for all cavities should be identical. Maintain an equal pressure drop through all gates to ensure acceptable parts. Use Y-splits instead of T-splits to help prevent bubbles in your part (see figure 11-1). A multiple-cavity mold having different-sized cavities may be filled through a single mix-head system with balanced runners.

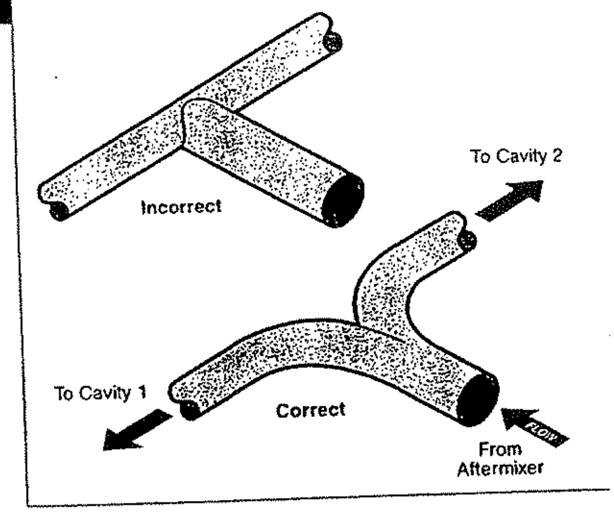
When using foamed systems in a multiple-cavity mold, pay special attention to the mold tilt. You may also have to use adjustable restrictors in the runner branches to control the degree of filling in individual cavities to ensure equivalent filling levels. Contact your Bayer representative for assistance on your specific application.

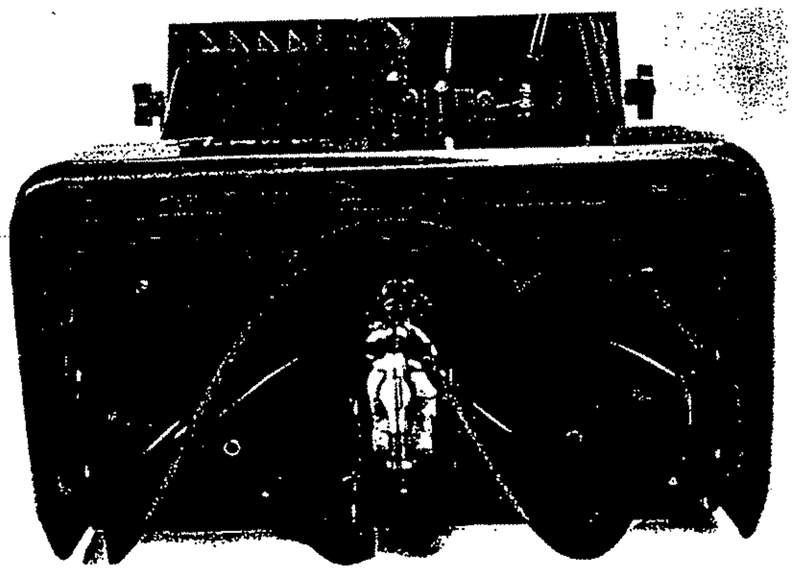


SELF-CONTAINED MOLDS

Self-contained molds do not require a press. Typically, these molds are more expensive than the mold in a mold-and-press setup, but generally cost less than a traditional mold and press. They are often used for simply shaped parts requiring low molding pressure. Contact your Bayer representative for more information on self-contained molds.

Figure 11-1 T-Split and Y-Split





When designing a mold, pay special attention to mold-construction materials, mold-surface treatments, and any textures or finishes to be applied to the mold. These topics are discussed in this section.

MOLD CONSTRUCTION MATERIALS AND FABRICATION TECHNIQUES

Because RIM polyurethanes generate heat when they react, you should choose a mold material that is conductive, to dissipate heat from the molding part. For this reason, metal molds are strongly suggested. Consider epoxy and spray-metal molds only for prototyping or low-volume production where cycle time and surface quality are of less importance. For high-volume runs, particularly those using reinforcing fillers, steel is usually the mold material of choice. If your mold needs mechanical or hydraulic knockouts, metal molds can accommodate these actuators much better than epoxy ones.

Generally, molds should be able to withstand 200 psi of pressure as a safety measure, even though typical molding pressures do not exceed 100 psi. Any number of materials can be used to make molds, including steel, aluminum, zinc alloys, copper alloys, and nickel. Determining which material is the best for your particular mold depends upon several parameters, such as:

- Number of parts to be made
- Surface finish requirements
- Time available for mold construction
- Part tolerances, dimensions, and shape
- Single- or multiple-cavity molds
- Mold cycle time and heat conduction
- Quality of parts to be made

MATERIAL SELECTION

Because surface textures will be closely duplicated, nonporous mold surfaces are essential. A smooth mold surface greatly improves part release. Pits, gouges, and other surface imperfections in the mold can lead to poor part release and breakage. As rules of thumb:

- The RIM system and production parameters influence the mold-construction material.
- Part geometry influences the mold-construction technique.

Steel

Because of their high degree of production reliability, machined steel molds are especially advantageous for mass-produced parts. They offer long mold life, can be outfitted with elaborate automated ejection systems, and are less likely to be scratched or damaged than softer materials. Widely used for automotive parts which require long production runs, steel molds last longer and may be costly. Typically, steel molds are used for parts made of short-fiber-filled materials or composites, as steel resists abrasion.

Aluminum

Offering lighter weight, good heat conductivity, and lower machining costs than steel, aluminum has long been the material of choice for mold makers specializing in polyurethane molds. It is softer than steel and may not be suitable for very long runs or for use with composite systems.

Zinc Alloys (Kirksite)

High-quality cast zinc molds offer excellent, nonporous surfaces. They are relatively heavy and require closely spaced cooling lines for accurate temperature control, because they are not as heat conductive as aluminum.

Nickel Shells

For high-quality surface reproductions, consider using nickel shells. These shells have a high surface hardness and offer good release characteristics. Small molds may not need back support for the shell. Mount larger molds in a steel or aluminum support frame and backfill with a casting material for structural rigidity. Cooling lines can be attached or plated onto the backside of the shell prior to mounting.

Epoxy Molds

Used mostly for short-run, prototype parts, epoxy molds have poor temperature control, are fragile, and can have surface roughness. They are poor heat conductors, sometimes causing parts to stick. Epoxy should be used only for short runs of prototype parts, when quality is not important and low cost is a primary concern. Foamed parts made in epoxy molds tend to have thin, non-uniform skin. Applying a spray-metal surface to an epoxy mold can eliminate some of these shortcomings.

Before filling an epoxy mold with a polyurethane system, carefully condition the surface with a reactant such as isocyanate. Before using any reactant, please check appropriate literature on proper use, storage, and personal safety equipment. This cleaning should remove any amine catalysts in the mold cavities. Waxing all cavity surfaces before the first molding may help the demolding process. If any reactive material remains on the mold surface, you may have difficulty removing your first prototype part.